

SmartPlant Instrumentation

Detailed Engineering User's Guide



Version 2009 SP2 (9.0.2)

October 2009/December 2010

DSPI2-PE-200003A-2-Updated



Copyright

Copyright © 1995-2009 Intergraph Corporation. All Rights Reserved.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation

P.O. Box 240000

Huntsville, AL 35813

Street address: 170 Graphics Drive, Madison, AL 35758

Terms of Use

Use of this software product is subject to the End User License Agreement ("EULA") delivered with this software product unless the licensee has a valid signed license for this software product with Intergraph Corporation. If the licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives licensee permission to print a reasonable number of copies of the documentation as defined in the applicable license agreement and delivered with the software product for licensee's internal, non-commercial use. The documentation may not be printed for resale or redistribution.

Warranties and Liabilities

All warranties given by Intergraph Corporation about equipment or software are set forth in the EULA provided with the software or applicable license for the software product signed by Intergraph Corporation, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license. No responsibility is assumed by Intergraph for the use or reliability of software on equipment that is not supplied by Intergraph or its affiliated companies. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Trademarks

Intergraph, the Intergraph logo, PDS, SmartPlant, SmartMarine, FrameWorks, I-Convert, I-Export, I-Sketch, IntelliShip, INtools, ISOGEN, MARIAN, SmartSketch, SPOOLGEN, SupportManager, and SupportModeler are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Microsoft and Windows are registered trademarks of Microsoft Corporation. MicroStation is a registered trademark of Bentley Systems, Inc. Other brands and product names are trademarks of their respective owners.

Contents

SmartPlant Instrumentation Detailed Engineering User's Guide	15
Wiring in SmartPlant Instrumentation	17
General Wiring Definitions	19
Principles of Wiring Operations in SmartPlant Instrumentation	19
Splitting the Wiring Design Among Several Teams	19
Creating a Wiring Scheme	21
Name and Sequence Uniqueness of Wiring Items	21
Apply Naming Conventions to Wiring Items	22
Associate a Symbol with an Item	23
Associate a Symbol with Multiple Items	23
Remove Trailing Separators From Wiring Item Names	24
Cables, Cable Sets, and Wires	25
Creating and Managing Cables	25
Define a Cable Configuration	26
Edit a Cable Configuration	27
Delete a Cable Configuration	28
Create a Cable	28
Create a Connector	29
Edit Cable Properties	30
Add a Cable Set	31
Add a Wire to a Cable Set	32
Create a Cable Without Sets and Wires	33
Apply a Reference Cable to Multiple Cables Without Sets and Wires	33
Apply a Reference Cable to One Cable Without Sets and Wires	34
Edit Wire Properties	34
Apply Regional Wire Colors to Project Cables	35
Conventional Panels	37
Create a Junction Box	37
Create a Marshaling Rack	38
Create a Cabinet	39
Create an Unassociated Conventional Device Panel	41
Create a DCS (Distributed Control System) Panel	42
Create a PLC (Programmable Logic Controller) Panel	43
Create a Panel with Terminals and Connectors	44
Enter Power Supply Data for Panels and Instrument Tags	45
Create a Rack	46
Create a Slot	47
Device Panels and Cables	49
Device Panels in SmartPlant Instrumentation	49
Principles of Creating Device Panels	50
Create an Instrument with an Automatically Configured Device Panel	51
Create an Instrument with a Manually Configured Device Panel	52
Create an Instrument Without a Device Panel	54
Create Device Panels and Cables in the Instrument Index Standard Browser View	55
Create Device Panels for Selected Instruments	55
Associate Instruments with Device Panels	57
Dissociate Device Panels from Instruments	57

Create a Device Cable	58
Rename Device Cables in Batch Mode.....	59
Wiring Equipment and Apparatus	61
Wiring Equipment	61
Customize and Use Wiring Equipment Categories	62
Add User-Defined Wiring Equipment Category Properties	63
Create a Wiring Equipment Item	64
Wiring Items Hierarchy Examples	65
Create an I/O Card	68
Create an I/O Termination	70
Customize the Controller/Processor List Header in I/O Card Properties	71
Associate an I/O card with an I/O Termination.....	72
Apparatus	73
Define an Apparatus Configuration	73
Create an Apparatus Group	75
Add an Apparatus to an Apparatus Group	76
Edit Apparatus Terminal Properties	76
The Plug-and-Socket Group Wizard	77
Modify the Internal Links of a Plug-and-Socket Group.....	79
Terminal Strips and Terminals	81
Terminal Strips	81
Define a Terminal Strip Configuration	81
Modify a Terminal Strip Configuration	83
Terminal Strip Configuration Examples.....	84
Create a Terminal Strip	85
Move a Terminal Strip to Another Parent Item.....	85
Add a Terminal to a Terminal Strip.....	86
Add a Group of Terminals to a Terminal Strip.....	87
Edit the Properties of a Terminal.....	87
Move a Terminal.....	88
Select a Graphic for a Terminal Connection	89
Making Connections in SmartPlant Instrumentation.....	91
Define a Connection Type	91
Connection Type Examples	93
Connect a Cable or a Cable Set to a Terminal Strip.....	94
Connect Device Cables to a Terminal Strip in Batch Mode	95
Connect a Wire to a Terminal Strip	96
Move Cables, Cable Sets, and Wires.....	97
Disconnect Cables, Cable Sets, and Wires.....	97
Add Jumpers to the Existing Wiring	98
Set Preferences for Automatic Naming of Jumpers	99
Remove a Jumper from the Existing Wiring.....	99
Assigning Wire Designation	99
Updating Connection Data	100
View Cable Connection Destinations	100
Display Adjacent Connections of a Cable	100
Display the Farthest Connection of a Cable.....	101
Examples of Farthest Connections of Cables	101
Trace a Signal in a Point-to-Point Wiring Diagram.....	102
Change the Font Size of Cable and Cable Set Names.....	103
Generate a Connection Report from the Connection Window.....	103
I/O Assignment.....	105
Set I/O Assignment Instrument Tag Preferences.....	105
Effect I/O Assignment for an Instrument	106

Effect I/O Assignment for a Segment	107
Assign an Instrument to a Channel	107
Assign a Segment to a Channel	108
Create a Channel	108
Create a Control System Tag	109
Create a Control System Tag in the Instrument Index Standard Browser View	109
Rename a Control System Tag	110
Couple an Instrument with a Control System Tag	110
Decouple an Instrument and a Control System Tag	110
Delete a Control System Tag	111
Filter the Display of Data in the I/O Assignment Window	111
Cancel I/O Assignment	112
Generate an I/O Tag Assignment Report	112
Control System Tag Propagation	113
View and Edit Channel Data	114
Find I/O Channels for Assignment	115
Cross Wiring	117
Perform Manual Cross Wiring	117
Perform Semiautomatic Cross Wiring	118
Set Cross Wiring Preferences	120
Perform Automatic Cross Wiring	120
Create a Cross Wiring Cable	122
Generate a Panel Strip Report from the Cross Wiring Window	122
Auto-Wiring	123
Defining and Executing Auto-Wiring Tasks	123
Flow of Activities for Auto-Wiring	124
Pre-Assign Junction Boxes to Device Panels	125
Auto-Wire Pre-Assigned Junction Boxes	126
Auto-Wire Two Panels	127
Auto-Wire Control Systems	128
Auto-Wire Control Systems Connected to Pre-Assigned Junction Boxes	130
Auto-Wiring Task Statuses	131
Change an Auto-Wiring Task Status	132
Signal Propagation	133
Possible Cases of Signal Propagation	133
Re-Propagate a Tag Signal	135
Force Signal Propagation	135
Propagate a Tag Signal Manually	135
Automatically Propagate a Signal from a Non-Wiring Instrument	136
Create a Local Tag Signal	137
Filter the Local Signal Dialog Box	137
Create a General Signal	138
Link Tag Signals to a General Signal	139
Multiplex Tag Signals	139
De-Multiplex Tag Signals	140
Intrinsic Safety	141
Create an Intrinsically Safe Circuit Type	141
Manage Intrinsic Safety Circuit Types	141
Calculate Intrinsic Safety	142
Define Tag Numbers for Intrinsic Safety Loop Calculation	143
Define Circuit Cables for Intrinsic Safety Loop Calculation	143
Calculate Intrinsic Safety for a Loop	144
Cable Routing	145
Cable Routing Flow of Activities	145

Set Cable Routing Options	146
Customize Cable Routing Terminology	147
Manage Standard Widths	147
Manage Standard Positions	147
Set the Maximum Number of Cables for the Standard Positions	148
Create a Routing Section	148
Edit a Routing Section	149
Associate a Routing Section with Cables	149
Edit a Cable Routing	150
Copy Routing Data to Another Cable	150
Append Routing Data to a Cable	151
Cable Drums	153
Create and Manage a Pulling Area	153
Associate Cables with a Pulling Area	154
Define Cable Drum Attributes for Cable Types	154
Manage Cable Drums Manually	154
Assign Cables to Drums Automatically (with Optimization)	155
Assign Cables to Drums Manually (Without Optimization)	156
Managing Panel Locations	157
Define a New Panel Location	157
Modify Panel Location Properties	157
Change the Location of a Panel	158
Dissociate a Panel from a Location	158
Delete a Panel Location	158
Cable Block Diagrams	159
Create a Cable Block Diagram	159
Add Cables and Objects to an Existing Cable Block Diagram	160
View an Existing Cable Block Diagram	161
Disconnect Cables from Items in a Cable Block Diagram	161
Associate a Cable with a Panel	161
Creating Wiring Equipment for Cable Block Diagrams	162
Filtering Items on a Cable Block Diagram	163
Foundation Fieldbus and Profibus	165
Foundation Fieldbus Design	165
Flow of Activities for Foundation Fieldbus Design	166
Preliminary Definitions for Foundation Fieldbus Common Tasks	169
Define Segment-Wide Parameters	170
Spur Cable Lengths Example	171
Working with Function Blocks	172
Create a User-Defined Function Block	172
Associate Function Blocks with Instrument Types	173
Manage the Function Blocks Supporting Table	174
Fieldbus Instruments Common Tasks	175
Working with the Fieldbus Tag Numbers Browser	175
Define Foundation Fieldbus and Profibus Instrument Type Profiles	176
Create Foundation Fieldbus and Profibus Instruments	176
Associate Function Blocks with Instruments	179
Fieldbus and Profibus Segments Common Tasks	181
Create a Foundation Fieldbus Segment	182
Edit the Properties of a Foundation Fieldbus Segment	182
Associate an Instrument with a Segment	183
Dissociate an Instrument from a Segment	183

Change Instrument / Segment Association	183
Delete a Fieldbus Segment	184
Enable a Function Block for I/O Assignment	184
Change the Fieldbus Tag Number List Browser View for Segments	184
Fieldbus Cables Common Tasks	185
Create a Home-Run Cable	185
Create a Spur Cable	186
Fieldbus Plug-and-Socket Boxes (Panels)	186
Managing Fieldbus Panels (Plug-and-Socket Boxes) Common Tasks	187
Create a Plug-and-Socket Device Panel	187
Create a Plug-and-Socket Junction Box	188
Edit the Properties of a Plug-and-Socket Device Panel	188
Edit the Properties of a Plug-and-Socket Junction Box	189
Working with Terminators	189
Managing Terminators Common Tasks	191
Create an External Terminator in a Plug-and-Socket Box	192
Create an External Terminator on a Terminal Strip	192
Create an Internal Terminator	192
Move an External Terminator to Another Port in a Plug-and-Socket Box	193
Terminator (Plug-and-Socket Box) - Segment Association	193
Terminator (Terminal Strip) - Segment Association	193
Fieldbus Bricks	194
Managing Fieldbus Brick Assemblies and Apparatuses Common Tasks	195
Create a Fieldbus Brick Assembly	195
Configure a Fieldbus Apparatus	197
Making Connections and Associations for Fieldbus	199
Define or Edit a Connector Type	199
Define or Edit a Connector Type Pin Configuration	200
Delete a Connector Type	200
Create a Connector	200
Assign Connector Pins to Cable Wires	201
Associate a Pin or a Terminal with a Channel	201
Delete a Connector	202
Connect a Home-Run Cable to a Plug-and-Socket Box	202
Associate a Segment with a Home-Run Cable	203
Managing Fieldbus I/O Cards and I/O Assignment Common Tasks	205
Create a Fieldbus I/O Card	205
Effect Fieldbus I/O Assignment	206
Fieldbus Validation	207
Fieldbus Validation Common Tasks	209
Generate a Validation Report for One Segment	209
Generate a Validation Report for Multiple Segments	210
Generate a Validation Report for Multiple Segments from the Fieldbus Tag Numbers Browser	210
Generate a Segment Wiring Report	210
Profibus Design	211
Flow of Activities for Profibus Design	211
Profibus Design Common Tasks	213
Create a Profibus DP Segment	214
Create a Profibus PA Segment	214
Associate Profibus PA Instruments with a PA Segment	214
Associate Profibus DP Instruments with a DP Segment	214
Create a DP-Compatible PLC or DCS Panel	215
Associate a PLC or DCS Panel with a Profibus DP Segment	215

Generate an Enhanced Profibus Layout Report	216
HART Instruments	217
Flow of Activities for HART Instrumentation	217
Define Instrument Type Profiles for HART Instruments.....	218
Create a HART Instrument	218
Create a Virtual Digital Tag.....	220
Create an I/O Card Compatible with HART Instruments	220
Effect I/O Assignment for HART Instruments	220
Conventional and Fieldbus Multi-Input Devices	221
Managing Multi-Input Devices.....	221
Create a Plug-and-Socket Multi-Input Device Panel	222
Create a Multi-Input Device Panel with Terminal Connections	223
Create a Multi-Input Instrument with Plug-and-Socket Connections for a Fieldbus Segment.....	223
Create a Multi-Input Instrument with Terminal Connections for a Fieldbus Segment	225
Create a Multi-Input Instrument with Plug-and-Socket Connections for a Serial Loop Interface	226
Create a Multi-Input Instrument with Terminal Connections for a Serial Loop Interface	227
Telecom Design	229
Flow of Activities for Telecom Design	229
Arrangement of Telecom Items in the Domain Explorer.....	230
Supporting Tables for Telecom.....	231
Working with Old Equipment.....	232
Telecom Devices and Cables	233
Set Telecom Device Type Profile Defaults.....	233
Create a Telecom Field Tag.....	234
Duplicate a Telecom Field Tag.....	234
Create a Telecom Conventional Field Device.....	235
Create a Telecom Plug-and-Socket Field Device	236
Create a Telecom Cable	236
Telecom Panels	237
Create a Telecom Junction Box	237
Create a Splice Panel.....	238
Create a Distribution Frame	238
Create a Patch Panel	239
Equipment Cabinets and Equipment Items	241
Create a PA Cabinet.....	241
Create an Amplifier	242
Add a Connector to an Amplifier	243
Add an Amplifier Channel to an Amplifier.....	243
Create a PABX Cabinet.....	244
Create a PABX Rack	245
Create a Switch	245
Add a Connector to a Switch	246
Add a Switch Port.....	246
Create a Switch Channel.....	247
Create and Associate a Telephone Number	247
Create an Unassociated Telephone Number.....	248
Associate a Telephone Number with a Channel	248

Create a Miscellaneous Cabinet	248
Create a Hub Cabinet.....	249
Create Hub Equipment.....	250
Create a Hub Connector.....	250
Add a Port to a Hub	251
Create an Intercom Cabinet	251
Add Intercom Equipment.....	252
Add an Intercom Connector	252
Loop Drawings Module	253
CAD Drawing Blocks in SmartPlant Instrumentation	253
Block Types	254
Loop Blocks	256
Shipped Block Files.....	257
Managing CAD Drawing Blocks Common Tasks	258
CAD Drawing Block Assignment Common Tasks	261
Associate a Loop Block	263
Associate Instrument Blocks Automatically (via Instrument Type)	263
Change the Block Assignment Method from Automatic to Manual	264
Associate Instrument Blocks Manually	265
Update Block Assignments	265
Copy Associated Blocks to Other Instrument Tags.....	266
Copy Automatically Assigned Blocks to Tags of Other Loops	266
Assign an Instrument Block to a Drawing Page	267
Dissociate Blocks from Instruments	267
Specify Border and Logo Blocks for Loop Drawings.....	268
Assign General Blocks to All Loop Drawings via Instrument Type	268
Loop Drawing Types and Drawing Properties	269
Loop Drawing Types and Drawing Properties Common Tasks	269
Update Paths for Existing CAD Drawings And Block Files	275
CAD Loop Drawing Generation	276
Loop Drawing Generation Methods.....	277
Steps in Loop Drawing Generation Using a CAD Application.....	278
CAD Settings for SmartPlant Instrumentation	278
Sample Loop Drawing (After Generation)	281
CAD Loop Drawing Generation Common Tasks	282
Apply Loop Drawing Generation Method	283
Generate CAD Loop Drawings.....	283
Show the Latest Generated CAD Drawing.....	285
Display CAD Loop Drawing Generation History.....	285
Delete CAD Loop Drawing Generation History	286
Clear Previous Macro Result Records	286
Loop Drawing Reports	286
Generate a Report of CAD Loop Drawing Generation Errors.....	287
Working with Macros in SmartPlant Instrumentation	287
SmartPlant Instrumentation Macro Structure	288
Loop Block Macro Syntax.....	289
Title Block Macro List	289
Wiring Module Macros - Conventions	293
Generate a Macro Report.....	295
Using Macros in CAD Blocks	295
Customizing Macro Definitions.....	297
Create a New Hook-Up Macro Definition	298

Delete a Hook-Up Macro Definition	298
Working with User-Defined Macro Functions	299
Standard Functions	300
Create a User-Defined Macro Function.....	302
Modify a User-Defined Macro Function.....	302
Delete a User-Defined Macro Function	303
Using External Macro Data Sources	303
Connect to an ODBC Profile	306
Connect to Database Platforms Other Than ODBC.....	306
MicroStation and SmartPlant Instrumentation	307
System Requirements.....	308
Set MicroStation for SmartPlant Instrumentation.....	308
Working with MicroStation	311
Create Tag Sets (MicroStation).....	311
Export Tag Sets (MicroStation)	312
Import a Tag Set (MicroStation)	312
Link a Tag to a Drawing Element (MicroStation).....	312
Create a Cell Library (MicroStation)	313
Fence Elements for a Cell (MicroStation).....	313
Define the Cell Origin (MicroStation).....	314
Define the Coordinates of the Cell Origin (MicroStation)	314
Add a Cell to the Cell Library (MicroStation)	314
Preview a Cell (MicroStation)	314
Display a Cell on the Desktop (MicroStation).....	315
Generating a Report Using MicroStation.....	315
Hook-Ups in SmartPlant Instrumentation	315
Hook-Ups Module Window	316
Hook-Up Items and Libraries	316
Hook-Up Item Libraries and Sub-Libraries	317
Workflow for Creating Hook-Up Items in the Reference Explorer.....	317
Managing Hook-Up Items and Libraries Common Tasks	318
Managing Hook-Ups Common Tasks	324
Hook-Up Drawing Generation.....	331
Shipped Enhanced Hook-Ups Symbols	333
Managing Hook-Up Drawings Common Tasks	333
Create a New Hook-Up Macro Definition.....	344
Delete a Hook-Up Macro Definition	344
Hook-Up Reports	345
Calculating the Total of Hook-Up Items for a Bill of Material.....	346
Hook-Up Reports Common Tasks	347
Instrument Maintenance	350
Access Rights in the Maintenance Module.....	351
Start the Maintenance Module	351
Starting the Maintenance Module From Any SmartPlant Instrumentation Module	351
Starting the Maintenance Module From an Instrument Index Browser View	352
Filter Maintenance Activities	352
Generate a Maintenance Report.....	353
Preventive Maintenance Preliminary Activities	353
Define a Preventive Maintenance Attachment	353

Define a Preventive Maintenance Task.....	354
Define a Preventive Maintenance Activity	354
Performing Preventive Maintenance.....	355
Perform Preventive Maintenance	356
Schedule Maintenance	356
Extend Existing Scheduling	357
Enter Preventive Maintenance Details	358
Performing Breakdown Maintenance	359
Perform Breakdown Maintenance	359
Issue a New Work Request	360
Opening a Work Request	360
Create a New Repair Form	361
Fill Out a Repair Form	362
Maintenance Event Records.....	362
Defining and Managing Maintenance Event Records	363
Set a Date Format for Maintenance Event Record Names.....	364
Create a Maintenance Event Record	364
Complete a Maintenance Event Record.....	365
Dimensional Data and Piping	365
Principles of the DDP Module	366
Additional Settings	366
Start the DDP Module	367
Preliminary Procedures.....	367
Define the Process Connection Class/Rating	367
Define a Process Connection End Preparation	367
Define a Dimensional Group.....	368
Define Dimensional Group Properties.....	368
Duplicate a Dimensional Group.....	369
Assign Dimensional Data Forms	369
Assign a PSR File to a Dimensional Data Form	369
Assign a Dimensional Data Form to a Dimensional Group.....	369
Associate an Instrument Type with a DDP Group.....	369
Vendor Data	370
Display and Access Vendor Data	370
Enter Vendor Data.....	371
Working Data	371
Display and Access Working Data	372
Edit Working Data.....	372
Piping Design Area.....	373
Copy Working Data to the Default Library.....	373
Copy Default Data to Working Data	373
Copy from Vendor Data to Working Data.....	374
Modify Working Data Status	375
Regenerate Dimensional Data Sheets	376
Manage Default Dimensional Data	376
Enter Default Data into the Default Library.....	376
Edit Default Dimensional Data	377
Customize the Default Library Display and Layout	377
Services and Reports.....	378
Generate a Dimensional Data Sheet.....	378
Track the Data Status History	378
Generate DDP Reports	378

Printing Dimensional Data Sheets.....	380
Print Empty Sheet Forms	381
Construction Module.....	383
Activate the Construction Module	384
Construction Flow of Activities	384
Definitions in the Construction Module	385
Installation Index Category Definitions	385
Edit an Installation Index Category.....	385
Edit an Installation Index	386
Delete an Installation Index	386
View and Modify Assigned Installation Indexes	386
Construction Options	387
Construction - Example Scenario	387
View Installation Indexes in the Installation Indexes Dialog Box	388
Add an Installation Index	389
Add an Installation Index	389
Add an Installation Index Category	390
Assign Installation Indexes to Items	390
Reports, Revisions and Archives	393
Generate Reports For An Installation Index Revision Reports	393
Construction Module Revisions	393
Maintaining a Revision Archive	394
View an Archived Installation Index.....	394
Edit an Archived Revision	395
Performing Calibrations.....	395
Calibration Module Window	396
Calibration Module Commands.....	396
Enter Tag Number Dialog Box	397
Starting the Calibration Module.....	397
Start the Calibration Module from the SmartPlant Instrumentation Window	397
Start the Calibration Module from the Instrument Index Standard Browser.....	397
Search for Tag Numbers in the Calibration Module.....	398
Calibration Forms.....	400
Upgrading the Standard Calibration Forms	400
Upgrade the Standard Calibration Forms.....	402
Upgrade Calibration Results	403
Calibration Preliminary Activities.....	403
Add Customized Calibration Forms to the Database	404
Create a Calibration Profile	405
Define Calibration Profile Settings.....	405
Associate Calibration Profiles with Instrument Types	405
Dissociate Profiles from Instrument Types.....	406
Manage Code Tables	406
Defining Test Equipment.....	407
Prerequisites for Creating a Test Equipment Tag	407
Create a Test Equipment Tag	408
Calibration Settings Activities.....	408
Define Tag Calibration Settings.....	409
Copy Existing Process Data	410
Define Alarm / Trip Settings	410

Define the Set Point.....	410
Calibration Data Entry Activities.....	411
Select a Calibration Profile	412
Enter Calibration Data	412
Select Test Equipment	413
View Calibration History	414
List of Calibration Reports	414
Generate Calibration Reports.....	416
Loop Error Calculation	416
Calibration Error Calculations.....	417
Set Tags for Loop Error Calculation	418
Calculate Loop Errors.....	419
Working with the Fluke Interface.....	420
Set up a Profile for the Fluke Calibrator	421
Map a Serial Port to Use the Fluke Calibrator on a Citrix Console	422
Download Data from SmartPlant Instrumentation.....	422
Upload Data from the Fluke Calibrator.....	424
Index	427

SECTION 1

SmartPlant Instrumentation Detailed Engineering User's Guide

The Detailed Engineering User's Guide contains information about various modules and advanced features that were not mentioned in the Basic User's Guide. These module and features include:

- Projects and As-Built Data
- Process Data
- Calculation module
- Specifications module
- Working with different interfaces
- Working in an integrated environment

SECTION 2

Wiring in SmartPlant Instrumentation

SmartPlant Instrumentation provides the means to efficiently create a comprehensive and detailed wiring design of your plant. You can create various types of wiring items that perform specific functions in your plant. You can define and manage the following:

- Panels belonging to a number of different categories, such as junction boxes, DCS and PLC panels, marshaling racks, and so forth
- Plug-and-socket boxes that support Foundation Fieldbus and Profibus systems
- Field devices and their cables
- Multi-input devices with either terminal or plug-and-socket connections
- Terminal strips and terminals
- Wiring equipment (I/O cards, fieldbus bricks, safety barriers, hubs, relays, and so forth)
- Cables based on shipped and user-defined configurations that include cable sets (pairs, triads, and so forth), wires, and connectors

After creating the required wiring items, you can effect I/O assignment and make the required connections. This will enable you to create a point-to-point wiring diagram and generate numerous wiring reports using the Enhanced Report Utility.

SECTION 3

General Wiring Definitions

This section deals with various general definitions in wiring.

In This Section

Principles of Wiring Operations in SmartPlant Instrumentation.....	19
Splitting the Wiring Design Among Several Teams.....	19
Creating a Wiring Scheme.....	21
Name and Sequence Uniqueness of Wiring Items.....	21
Apply Naming Conventions to Wiring Items	22
Associate a Symbol with an Item.....	23
Associate a Symbol with Multiple Items	23
Remove Trailing Separators From Wiring Item Names.....	24

Principles of Wiring Operations in SmartPlant Instrumentation

SmartPlant Instrumentation creates and keeps track of wiring routing and connections by identifying the wires and their connection points.

The wires and the items to which they are connected are arranged hierarchically as they appear in the **Domain Explorer**. These terms are described in detail in the relevant help topics.

You can make connections between or within any of the <unit> in a <plant>. Reference cables and panels are, however, available for all the <plants> in the domain.

The following sequence of operations is necessary for the creation of a wiring scheme:

- Creation of panels, plug-and- socket boxes, terminal strips, wiring equipment, terminals, and so on
- Creation of cables, cable sets, and wires
- Connecting wires to terminals

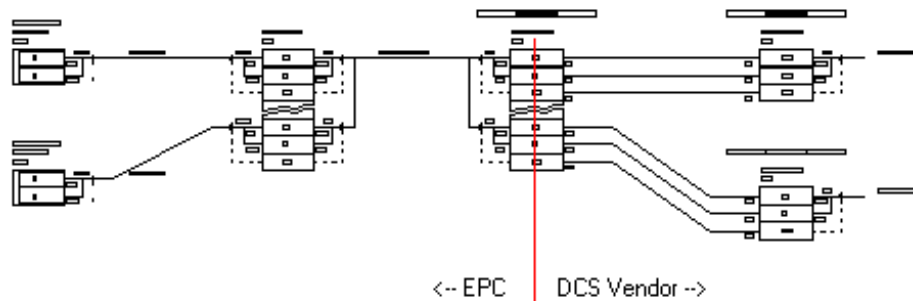
In addition, tagged instruments may be wired so that their wiring routing is automatically tracked and wire names are automatically propagated. For details, see:

- *Device Panels* (see "*Device Panels in SmartPlant Instrumentation*" on page 49)
- *I/O Assignment* (on page 105)

Splitting the Wiring Design Among Several Teams

There are projects in which, for various reasons, several teams work on the wiring design of the same project. These teams can be located at the same site or be in different geographical locations.

A good example of such a project is when the main contractor designs the field instrumentation wiring up to the marshaling racks while the DCS vendor is responsible for the DCS design and also for the design of the wiring from the marshaling panels to the terminals of the I/O cards that they supply, as shown below:



At the end of the project, or at any intermediate stage, there is a need to integrate both designs into one database to complete the wiring from the field to the DCS and then generate the complete loop diagrams and the wiring reports.

At this time, the boundary or the scope of the design that each party is responsible for can vary from project to project. Usually, the intersection point would be somewhere around a marshaling panel, either at the field side of a terminal strip and the control system terminal strip side. This issue is usually discussed between the parties and they agree where to draw this line.

In SmartPlant Instrumentation, we recommend the following strategy to handle this split wiring design:

1. The main contractor or the EPC starts with the creation of an As-Built domain and then creates a project for each of the working parties involved in the overall design. In the example mentioned above, the main contractor works on the main As-Built plant and defines the projects for the DCS vendors.
2. All the teams work on the same database using Citrix. However, it is also possible to work offline.

Using this approach for splitting the design has the following advantages:

- Provides the ability to see the current As-Built data while connecting to the data (and not the other way around).
- Enables good control of new tag numbers and subcomponents.
- Retains the uniqueness of rules when adding various subcomponents such as terminals on a terminal strip.
- Enables the comparison of As-Built loops and project loops.

Note that the members of the various teams working on the projects have to be informed of the changes made in As-Built so that they can update their databases. Consequently, based on the data maturity, teams should coordinate among themselves when to transfer their wiring data to the rest of the projects.

For more information, see Owner Operator Domain (As-Built and Projects) in the *Administration User's Guide*, under *Domain and Project Administration*, *Owner Operator: General Activities*.

Creating a Wiring Scheme

After the System Administrator initializes and creates a domain, SmartPlant Instrumentation creates the following items:

- Reference cables
- Reference device panels with terminal strips
- Connection types

These reference items suffice for the most common connections to device panels. For more sophisticated combinations, you should create your own reference items.

For example, for a 2-wire device you need to create in the **Reference Explorer**:

- A device panel with a two- terminal strip named DEFAULT 2W DEVICE PANEL
- A cable with a shielded twisted pair (cable + one shielded pair), named DEFAULT 2W CABLE
- A connection device that connects two wires from a pair one after the other named 2 IN A ROW

You can then select the above items the wiring profile to define the reference wiring properties for a given instrument type.

When you start a new wiring scheme, the folders in the **Domain Explorer** are empty. You can right-click one of the panel folders in the **Domain Explorer** and on the shortcut menu, click **New** and then click the required option. The **Reference Explorer** contains several reference panels and cables that are provided as shipped data that you can copy to the **Domain Explorer**.

NOTE You must have appropriate access rights to alter the reference items.

Name and Sequence Uniqueness of Wiring Items

When creating or editing the properties of a wiring item, SmartPlant Instrumentation validates that the wiring item has a unique name and sequence. The software applies the following validation rules:

Item Sequence

If SmartPlant Instrumentation detects that the sequence of a wiring item is not unique, the software displays an appropriate message and does not create the item until you enter a unique sequence.

The software validates sequence uniqueness according to the following rules:

- In an owner operator domain, the software validates sequence uniqueness in As-Built and all the projects belonging to the current owner operator domain.
- Panels, cables, and connectors do not require any sequence.
- Terminals must have a sequence which is unique at the terminal strip level.
- All the other wiring items must have a sequence which is unique at the level of the immediate parent item under which the item exists.

Item Names

If SmartPlant Instrumentation detects that an item does not have a unique name, the software displays a warning message, after which you proceed depending on the setting you made on the **General** page of the wiring preferences. You can set the software to proceed without changing the item name if you do not select the **Enforce name uniqueness** wiring preference option on the **General** page. If you select the **Enforce name uniqueness** check box, you must rename the item in order to proceed with your work.

The software validates name uniqueness according to the following rules:

- In an owner operator domain, the software validates name uniqueness in As-Built and all the projects belonging to the current owner operator domain.

- For panels and cables, SmartPlant Instrumentation checks for name uniqueness at the level of the current <plant>.
- Wires do not require name uniqueness.
- The software checks for terminal name uniqueness at the terminal strip level.
- For all the other wiring items, SmartPlant Instrumentation checks for name uniqueness at the level of the immediate parent item under which the item exists.
- If you try to create a wiring item that has no name, the software prompts you to enter a name. If you select the **Enforce name uniqueness** wiring preference option, the software treats items without names according to the validation rules stated above. If you do not select this wiring preference option, the software allows you to create wiring items without names.
- When moving an item to another parent item, the software validates the name uniqueness at the target level. If you did not select the **Enforce name uniqueness** wiring preference option, the software moves the item and retains its name. If you selected the **Enforce name uniqueness** wiring preference option, the software does not allow you to move the item if the item name violates the validation rules.

For more information, see *Naming Conventions* in the *Administration User's Guide*, under *Domain and Project Administration, Naming Conventions*.

Apply Naming Conventions to Wiring Items

SmartPlant Instrumentation allows you to apply naming conventions to various wiring items. You can apply naming conventions to all panels, racks, and wiring equipment. Note that naming conventions are set by your Domain Administrator.

After applying the naming conventions, the software renames the selected items according to the existing conventions. You can apply naming conventions to a single item or multiple items. If you need to that for a single wiring item, you can do it in the **Properties** dialog box of that item by selecting the **Apply naming convention** check box.

1. In the **Domain Explorer**, do one of the following:
 - In the **Tree view**, select one desired item.
 - Expand a folder and in the **Items** pane, select the items you require.
2. Right-click the selected items and then on the shortcut menu, click one of the following:
 - **Apply Naming Conventions to Selected Items** — to rename only the items you selected without renaming the child items that exist on the lower hierarchy level of these items.
 - **Apply Naming Conventions to Selected Items and Child Items** — to rename the items you selected and all the child items that exist on the lower hierarchy level of these items.

Associate a Symbol with an Item

IMPORTANT

- Make sure that you have defined a custom symbol folder in the SmartPlant Instrumentation **Preferences** dialog box for the appropriate enhanced report type before you associate a symbol with an item.
 - When you associate a symbol with an item that can have more than one possible relation to other items for which you want to display data, make sure that the symbol you select uses macros with the appropriate relations. If a macro has an incorrect relation, the software cannot display the data.
1. In the **Domain Explorer**, select the desired item, for example, a particular panel.
 2. Right-click the item and click **Properties**.
 3. Click the **Associate Symbols** tab.
 4. Click **New**.
 5. In the **Name** column, select the desired report type.
 6. Beside the **Symbol File** column, click **Browse**.
 7. Navigate to the desired .sym file.
 8. If you require, repeat steps 4 -7 to associate symbols with the item for other report types.

TIP When associating symbols with a panel, for example, you will use one symbol to represent the panel in a Location Layout report, and another symbol to represent the panel in a Panel Layout report.

NOTES

- For an instrument, when you associate a symbol for Enhanced Report Utility reports, the symbol will also be displayed in other wiring reports.
- You can associate each item with one symbol only per available report type.
- You can associate the same symbol with more than one item if you desire.
- When using custom terminal symbols in enhanced reports, there are two ways of displaying the terminals on the drawing sheet:
 - Associating a symbol with a panel, where the symbol exactly represents the panel-strip-terminal configuration, and dragging the panel onto the drawing sheet. With this method, the software displays the entire panel.
 - Associating a symbol with each individual terminal and dragging the desired terminals onto the drawing sheet. With this method, the software displays the selected terminals with their associated strips and panels, together with the appropriate macros.

Associate a Symbol with Multiple Items

You can associate a symbol with multiple items using the **Symbols Batch Association** command.

IMPORTANT

- Make sure that you have defined a custom symbol folder in the SmartPlant Instrumentation **Preferences** dialog box for the appropriate enhanced report type before you associate a symbol with an item.
- When you associate a symbol with an item that can have more than one possible relation to other items for which you want to display data, make sure that the symbol you select uses macros with the appropriate relations. If a macro has an incorrect relation, the software cannot display the data.

1. In the Domain Explorer click on the folder that contains the items you want to associate with a symbol. For example, **Device Panels**.
2. In the **Items** pane of the **Domain Explorer**, select the items you want to associate with a symbol.
3. Right-click on the selected items, and from the shortcut menu select **Actions > Symbols Batch Association**.
4. On the **Symbols Batch Association** dialog box, click **New**.
5. In the **Name** column, select the desired report type.
6. Beside the **Symbol File** column, click **Browse**.
7. Navigate to the required symbol.
8. Click **OK**.

NOTE The **Symbols Batch Association** command is also available in various browsers using the browser shortcut menu > browser category, for example **Wiring** or **Instrument Index > Symbols Batch Association**.

Remove Trailing Separators From Wiring Item Names

Wiring items that have been created according to naming conventions sometimes may have trailing separators. This happens when the last naming convention segment is not defined and there is no value for it. You can instruct the software to remove the trailing separators automatically when creating wiring items.

1. In the Wiring module, click **File > Preferences**.
2. In the **Preferences** dialog box, under **Wiring**, click **Custom**.
3. In the **Custom features for the Wiring module** group box, under **Parameter**, type **Remove Trailing Separator**.
4. Under **Value** type **Y**.

NOTE The default setting is **N** which instructs the software not to remove the trailing separators.

SECTION 4

Cables, Cable Sets, and Wires

This section describes the creation and management of cables, cable sets, and wires.

In This Section

Creating and Managing Cables	25
Define a Cable Configuration.....	26
Edit a Cable Configuration.....	27
Delete a Cable Configuration.....	28
Create a Cable.....	28
Create a Connector	29
Edit Cable Properties.....	30
Add a Cable Set.....	31
Add a Wire to a Cable Set	32
Create a Cable Without Sets and Wires	33
Apply a Reference Cable to Multiple Cables Without Sets and Wires	33
Apply a Reference Cable to One Cable Without Sets and Wires..	34
Edit Wire Properties	34
Apply Regional Wire Colors to Project Cables	35

Creating and Managing Cables

You create new cables in SmartPlant Instrumentation based on a particular cable configuration. A cable configuration includes a particular arrangement of cable sets (pairs, triads, and so forth within a cable) and wires to be built with the cable. Therefore, when creating a new cable, you have to select an existing cable configuration or define a new one. After creating a cable, you can add additional cable sets and wires to that cable if required.

As a time saver, it is recommended that you create reference items for the cable, cable set, and wire arrangements you intend to use frequently. You create these reference cables in the **Reference Explorer**. You can then drag them to the **Domain Explorer** and this way create numerous cables on the fly.

In **SmartPlant Instrumentation Explorer**, cables occupy the top hierarchy level under which you can create various wiring items such as cable sets, wires, and connectors. There is a lot of flexibility in creating a cable hierarchy. There is no rigid structure like "cable – cable set – wire" that limits your cable design. You can create various cable hierarchy structures, as you require. See Cable Hierarchy Example to examine a possible structure.

Define a Cable Configuration

This option allows you to create a new cable configuration. All SmartPlant Instrumentation cables are based on a particular cable configuration. Therefore, you need a cable configuration to create a new cable. A cable configuration includes a cable set (pairs, triads, and so forth within a cable) and a wire arrangement. As a result, when creating a new cable, SmartPlant Instrumentation automatically creates the required cable sets and wires within that cable based on the configuration that you define. You can add more cable sets and wires to that cable manually later if needed.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. In the **Domain Explorer** or **Reference Explorer**, right-click the **Cables** folder and then click **New > Cable**.
3. In the **Cable Configuration** dialog box, click **New**.
4. In the **New Cable Configuration** dialog box, enter the total number of sets (pairs, triads, and so forth) that SmartPlant Instrumentation will create automatically when a new cable is based on this new configuration.
TIP Add an additional cable set for an overall shield.
5. From the **Default cable set type** list, select the appropriate cable type.
TIP The selected cable type determines whether cables based on this configuration have multiple wires, pairs, triads, and so forth.
6. Click **OK** to return to the **Cable Configuration** dialog box.
7. In the **Cable Configuration** dialog box, type the name of the new configuration in the **Cable Configuration** data field.
8. In the **Cable default name** field, type a name of the cables that will be based on the current configuration.
TIP This means that every time you create a new cable based on this new configuration, SmartPlant Instrumentation uses this default setting to name the new cable. You can then modify this name as needed when you define the cable in the **Cable Properties** dialog box.
9. In the **Description** text field, type a brief description of the new configuration.
10. Click in the **Cable Set** text field and type the name of each cable set that is defined in this configuration. You can rename the individual cable sets later if needed when editing the cable set in the **Cable Set Properties** dialog box.
11. For each cable set, in the **Cable Set Type** field, accept the displayed value (the cable set type you selected in the **New Cable Configuration** dialog box) or select another one if needed.
12. For each first cable set of a given cable set type, do the following under **Cable set wire details**:
 - a. Click in the **Wire Tag** text field and type the names of the individual wires within the cable sets. You can accept the displayed value if needed.
 - b. From the **Wire Color** list, select a color for each wire. If the required value is not available, you can add it in the **Wire Color** dialog box that you access from the **Tables** menu in the **Wiring Module** window.
 - c. From the **Wire Type** list, select a type for each wire. If the required value is not available, you can add it in the **Wire Type** dialog box that you access from the **Tables** menu in the **Wiring Module** window.
 - d. From the **Polarity** list, select the appropriate polarity or shield for each wire.

- e. For additional cable sets based on a cable set type for which you already entered settings under **Cable set wire details**, do one of the following:
 - To copy the values from a selected cable set to all sets based on the current cable set type, click **Apply**. You can select the **Exclude wire tags** check box before you click **Apply** if you do not wish to copy the wire tags while copying wire color, wire type, and polarity details.
 - Enter the settings individually for each cable set.
13. Click **Save**.

Edit a Cable Configuration

This procedure explains how to modify an existing cable configuration.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. In the **Domain Explorer** or **Reference Explorer**, right-click the **Cables** folder and then click **New > Cable**.
3. In the **Cable Configuration** dialog box, select a configuration from the **Cable configuration** list.
4. Click **Edit**.
5. In the **Cable Configuration** dialog box, enter the total number of sets (pairs, triads, and so forth) that SmartPlant Instrumentation will create automatically when a new cable is based on this new configuration.

TIP Add an additional cable set for an overall shield.
6. From the **Default cable set type** list, select the appropriate cable type.

TIP The selected cable type determines whether cables based on this configuration have multiple wires, pairs, triads, and so forth.
7. Click **OK** to return to the **Cable Configuration** dialog box.
8. In the **Cable Configuration** dialog box, type the name of the new configuration in the **Cable Configuration** data field.
9. In the **Cable default name** field, type a name of the cables that will be based on the current configuration.

TIP This means that every time you create a new cable based on this new configuration, SmartPlant Instrumentation uses this default setting to name the new cable. You can then modify this name as needed when you define the cable in the **Cable Properties** dialog box.
10. In the **Description** text field, type a brief description of the new configuration.
11. Click in the **Cable Set** text field and type the name of each cable set that is defined in this configuration. You can rename the individual cable sets later if needed when editing the cable set in the **Cable Set Properties** dialog box.
12. For each cable set, in the **Cable Set Type** field, accept the displayed value (the cable set type you selected in the **New Cable Configuration** dialog box) or select another one if needed.
13. For each first cable set of a given cable set type, do the following under **Cable set wire details**:
 - a. Click in the **Wire Tag** text field and type the names of the individual wires within the cable sets. You can accept the displayed value if needed.
 - b. From the **Wire Color** list, select a color for each wire. If the required value is not available, you can add it in the **Wire Color** dialog box that you access from the **Tables** menu in the **Wiring Module** window.

- c. From the **Wire Type** list, select a type for each wire. If the required value is not available, you can add it in the **Wire Type** dialog box that you access from the **Tables** menu in the **Wiring Module** window.
 - d. From the **Polarity** list, select the appropriate polarity or shield for each wire.
 - e. For additional cable sets based on a cable set type for which you already entered settings under **Cable set wire details**, do one of the following:
 - To copy the values from a selected cable set to all sets based on the current cable set type, click **Apply**. You can select the **Exclude wire tags** check box before you click **Apply** if you do not wish to copy the wire tags while copying wire color, wire type, and polarity details.
 - Enter the settings individually for each cable set.
14. Click **Save**.

Delete a Cable Configuration

This procedure explains how to delete a cable configuration.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. In the **Domain Explorer** or **Reference Explorer**, right-click the **Cables** folder and then click **New > Cable**.
3. In the **Cable Configuration** dialog box, select a configuration from the **Cable configuration** list.
4. Click **Delete**.

Create a Cable

You can create new cables either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a cable that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference cables facilitate fast creation of plant cables by copying an existing cable configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a cable that does not have a configuration that you intend to use frequently, we recommend that you create this cable in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the cables in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. In the **Domain Explorer** or **Reference Explorer**, right-click the **Cables** folder and then click **New > Cable**.
3. In the **Cable Configuration** dialog box, select a cable configuration from the **Cable configuration** list.

TIP Click **New** if you do not have a suitable configuration. For details, see *Define a Cable Configuration* (on page 26).
4. Click **Create**.
5. In the **Cable Properties** dialog box, under **Cable**, type the cable name.

TIP If you are working under cable type dependency, your selection of cable type determines the reference cable name.

6. Under **Description**, type the new cable description as needed.
7. Select one of the following under **Cable class**:
 - To create a conventional cable, accept the default setting **Conventional**.
 - To create a fieldbus spur, select **Conventional**.
 - To create a fieldbus home-run cable, select **Fieldbus home-run**.
 - To create a telecom cable, select **Telecom**.
8. Under **Type**, select the required cable type if needed. If the required type is not available, click  next to the list arrow to add the required item to the list.
9. Select the cable manufacturer, model, color as appropriate. If the item that you need is not on the list, click .
10. Type the end one, and end two locations of the cable.
11. Type the cable length and select the unit of measure.
12. To assign this cable to a cable drum, select values from the **Pulling area** list and from the **Cable drum** list. For details, see *Assign Cables to Drums Manually (Without Optimization)* (on page 156).
13. To define a cable harness, select the cable harness from the list. If the item that you need is not on the list, click .
14. To name the wires ends in this cable according to a naming convention, select the appropriate naming convention from the **Wire end naming convention** list.
15. To define the cable as intrinsically safe, select **Set as intrinsically safe**.
16. To define glands for the cable, from the **Glands** group box, select the **End 1**, **End 2** or both cable glands. If the required gland is not available, click .
Note the graphic representation of cable glands in the software:
 -  — A cable with no glands
 -  — A cable with a gland on one of its sides
 -  — A cable with a gland on both sides
17. Click **Connectors** to create connectors for the cable and manage connector-wire assignments.
18. In the **Cable Properties** dialog box, click **OK**.

Create a Connector

Connectors are needed for fieldbus home-run cables, telecom cables, and other cables that require connectors. SmartPlant Instrumentation allows you to build your wiring system with any of the following connector implementations:

- Connector at one or both cable ends
 - Connector at end of a single cable set
 - Connector at end of wires from different cable sets
1. In the **Domain Explorer** or **Reference Explorer**, do one of the following:
 - To create a new cable, right-click the **Cables** folder and on the shortcut menu click **New > Cable**.
 - Right-click an existing cable, and on the shortcut menu click **Properties**.
 2. Modify cable properties as needed. For details, see *Create a Cable* (on page 28).

3. Click **Connectors**.
4. In the **Connector Assignments** dialog box, do one of the following:
 - To manage connector-wire associations for end 1 of the cable, click the **End 1** tab.
 - To manage connector-wire associations for end 2 of the cable, click the **End 2** tab.
5. In the **Connector Assignments** dialog box, click **New**.
6. In the **Connector Properties** dialog box that opens, under **Connector**, type the name of the new connector.
 - From the list, select a connector type as configured.
 - To modify or create a connector type, click . For details, see *Define or Edit a Connector Type* (on page 199).
7. Under **Male/female** select **Male** or **Female** as needed.
8. Click **OK** to return to the **Connector Assignments** dialog box.

Edit Cable Properties

This procedure shows you how to edit the properties of a cable.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Expand the **Cables** folder and right-click a cable.
3. On the shortcut menu, click **Properties**.
4. In the **Cable Properties** dialog box, under **Cable**, type the cable name.

TIP If you are working under cable type dependency, and have already created plant cables based on a reference cable that you are editing, certain reference cable fields are not enabled for editing. The System Administrator can enable cable type dependency when making domain definitions.
5. Under **Description**, type the new cable description as needed.
6. Select one of the following under **Cable class**:
 - To create a conventional cable, accept the default setting **Conventional**.
 - To create a fieldbus spur, select **Conventional**.
 - To create a fieldbus home-run cable, select **Fieldbus home-run**.
 - To create a telecom cable, select **Telecom**.
7. Under **Type**, select the required cable type if needed. If the required type is not available, click  next to the list arrow to add the required item to the list.
8. Select the cable manufacturer, model, color as appropriate. If the item that you need is not on the list, click .
9. Type the end one, and end two locations of the cable.
10. Type the cable length and select the unit of measure.
11. To assign this cable to a cable drum, select values from the **Pulling area** list and from the **Cable drum** list. For details, see *Assign Cables to Drums Manually (Without Optimization)* (on page 156).
12. To define a cable harness, select the cable harness from the list. If the item that you need is not on the list, click .

13. To name the wires ends in this cable according to a naming convention, select the appropriate naming convention from the **Wire end naming convention** list.
14. To define the cable as intrinsically safe, select **Set as intrinsically safe**.
15. To define glands for the cable, from the **Glands** group box, select the **End 1**, **End 2** or both cable glands. If the required gland is not available, click .
Note the graphic representation of cable glands in the software:
 -  — A cable with no glands
 -  — A cable with a gland on one of its sides
 -  — A cable with a gland on both sides
16. Click **Connectors** to create connectors for the cable and manage connector-wire assignments.
17. In the **Cable Properties** dialog box, click **OK**.

Add a Cable Set

SmartPlant Instrumentation adds cable sets (pairs, triads, and so forth within a cable) automatically when you create a new cable based on a cable configuration. This procedure describes how you can add another cable set to an existing cable that already contains a number of cable sets.

IMPORTANT If you are working under cable type dependency, and have already created plant cables based on a particular reference cable, the software does not allow you to add cable sets to such a reference cable. Further, you cannot add cable sets to a plant cable created under cable type dependency. The System Administrator can enable cable type dependency when making domain definitions.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Expand the **Cables** folder and right-click a cable.
3. On the shortcut menu, click **New > Cable Set**.
4. In the **Cable Set Properties** dialog box, type the cable set name in the **Cable set** text field.
5. Under **Cable set sequence**, enter the sequence of the cable set in the cable.

TIP Typically, the cable set sequence is determined by the manufacturer. If you change the sequence number of a cable set, make sure that you also change the sequence number of the other cable sets in the cable as appropriate to avoid duplicate values and erroneous terminations.
6. Click **OK** to add a new cable set.

Add a Wire to a Cable Set

SmartPlant Instrumentation adds wires automatically when you create a new cable based on a cable configuration. This procedure describes how you can add another wire to an existing cable set. You can add a wire either to a cable set created automatically or to a cable set that you added manually.

IMPORTANT If you are working under cable type dependency, and have already created plant cables based on a particular reference cable, the software does not allow you to add cable sets or wires to such a reference cable. Further, you cannot add cable sets or wires to a plant cable created under cable type dependency. The System Administrator can enable cable type dependency when making domain definitions.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Expand the **Cables** folder and then double-click a cable to display its cable sets.
3. Right-click a cable set and then on the shortcut menu, click **New > Cable Wire**.
4. In the **Wire Properties** dialog box, under **Wire**, type the wire tag.

TIP By default, wire tags are based on the signal that they carry. To name wires based on user input the Domain Administrator needs to enable cable type dependency when making domain definitions.
5. In the **Details** group box, under **Wire sequence in set**, accept the automatically assigned value that determines the numbering sequence of the wire in the cable set.

TIP The values that SmartPlant Instrumentation suggests under **Wire sequence in set** are unique throughout As-Built and all engineering projects where the current cable set appears.
6. From the **Polarity** list, select the signal polarity or shield.

TIP When you create a cable for the first time, or when you copy from reference cables, check for the existence of shield indication wherever needed. Shield indication is important when SmartPlant Instrumentation propagates signals through wires. In this case, the software does not overwrite the wire tag with the tag signal. You can also define a wire as an overall shield if required.
7. From the **Type** list, select the required wire type. If the required type is not on this list, click  next to the list arrow to open a dialog box where you can add, modify, or delete items from the **Wire type** list.
8. From the **Color** list, select the required wire color. If the required color is not on this list, click  next to the list arrow to open a dialog box where you can add, modify, or delete items from the **Wire color** list.
9. In the **Note** text field, type a brief note if needed.
10. To name the wire ends according to a naming convention, select the appropriate naming convention from the **Wire end naming convention** list. Note that this check box is available only if the Domain Administrator has enabled it in the **Wire End Naming Conventions** dialog box.
11. Do the following to name the wire ends not according to a naming convention:
 - a. Select **Override naming convention**.
 - b. Under **Wire end 1**, type the first wire end name.
 - c. Under **Wire end 2**, type the second wire end name.
 - d. Click **Revisions** to manage revisions for the current wire.

12. Click **OK** in the **Wire Properties** dialog box.

Create a Cable Without Sets and Wires

This option allows you to create a cable that does not have any cable sets or wires. You can use such cables in the early stages of your design and when creating cable block diagrams where it is not necessary to specify cable types, the number of sets and wires in the cables, and so forth.

IMPORTANT The software does not allow you to create cables without sets and wires if you are working under cable type dependency.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Right-click the **Cables** folder and then on the shortcut menu click **New > Cable**.
3. On the **Cable Configuration** dialog box, click **New**.
4. Do the following on the **New Cable Configuration** dialog box:
 - a. Under **Number of cable sets**, type **0** (zero).
 - b. Click **OK**.
5. On the **Cable Configuration** dialog box, click **Save** and then **Create**.
6. On the **Cable Properties** dialog box, enter the cable properties as you require.

TIP Since cables without cable sets and wires are useful during the early stages of plant design and when creating cable block diagrams, the only cable property that you need to enter is the cable name. You can populate the other properties when you apply a reference cable to this cable.

7. Click **OK**.

Apply a Reference Cable to Multiple Cables Without Sets and Wires

This feature enables you to apply a reference cable to cables that do not have any sets or wires. This batch operation enables you to select multiple cables in the **Domain Explorer** and then select an appropriate reference cable for each cable that you selected.

1. In the **Domain Explorer**, click the **Cables** folder.
2. In the **Items** pane, press and hold the Ctrl key and then right-click the cables you require.
3. On the shortcut menu, click **Actions > Apply Reference Cable**.
4. On the **Apply Reference Cable** dialog box, select an appropriate reference cable for each cable.
5. Click **OK**.

Apply a Reference Cable to One Cable Without Sets and Wires

This procedure explains how to apply a reference cable to a single cable that does not have any sets or wires. You can apply a reference cable or a cable structure as you require.

1. In the **Domain Explorer**, click the **Cables** folder.
2. In the tree view right-click a cable you require.
3. On the shortcut menu, click **Actions > Apply Cable Structure**.
4. On the **Apply Cable Structure** dialog box, do one of the following:
 - Click the **Create items based on a reference cable** option button and then select an appropriate reference cable from the list.
 - Click the **Create items based on a cable configuration** option button and then select an appropriate cables structure from the list. You can click  to open the **Cable Configuration** dialog box to define a new cable configuration if needed.
5. Click **OK**.

Edit Wire Properties

This option describes how to edit the properties of a wire.

IMPORTANT If you are working under cable type dependency, and have already created plant cables based on a particular reference cable, the software does not allow you to add cable sets or wires to such a reference cable. Further, you cannot add cable sets or wires to a plant cable created under cable type dependency. The System Administrator can enable cable type dependency when making domain definitions.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Expand the **Cables** folder and then double-click a cable to display its cable sets.
3. Double-click a cable set to display the existing wires.
4. Right-click a wire and then on the shortcut menu, click **Properties**.
5. In the **Wire Properties** dialog box, under **Wire**, type the wire tag.

TIP By default, wire tags are based on the signal that they carry. To name wires based on user input the Domain Administrator needs to enable cable type dependency when making domain definitions.
6. In the **Details** group box, under **Wire sequence in set**, accept the automatically assigned value that determines the numbering sequence of the wire in the cable set.

TIP The values that SmartPlant Instrumentation suggests under **Wire sequence in set** are unique throughout As-Built and all engineering projects where the current cable set appears.
7. From the **Polarity** list, select the signal polarity or shield.

TIP When you create a cable for the first time, or when you copy from reference cables, check for the existence of shield indication wherever needed. Shield indication is important when SmartPlant Instrumentation propagates signals through wires. In this case, the software does not overwrite the wire tag with the tag signal. You can also define a wire as an overall shield if required.

8. From the **Type** list, select the required wire type. If the required type is not on this list, click  next to the list arrow to open a dialog box where you can add, modify, or delete items from the **Wire type** list.
9. From the **Color** list, select the required wire color. If the required color is not on this list, click  next to the list arrow to open a dialog box where you can add, modify, or delete items from the **Wire color** list.
10. In the **Note** text field, type a brief note if needed.
11. To name the wire ends according to a naming convention, select the appropriate naming convention from the **Wire end naming convention** list. Note that this check box is available only if the Domain Administrator has enabled it in the **Wire End Naming Conventions** dialog box.
12. Do the following to name the wire ends not according to a naming convention:
 - a. Select **Override naming convention**.
 - b. Under **Wire end 1**, type the first wire end name.
 - c. Under **Wire end 2**, type the second wire end name.
13. Click **Revisions** to manage revisions for the current wire.
14. Click **OK** in the **Wire Properties** dialog box.

Apply Regional Wire Colors to Project Cables

This procedure allows you to apply regional wire colors to your project cables based on reference cables.

IMPORTANT This feature works only if your System Administrator has enabled the cable type dependency when making domain definitions.

1. Import the required reference cables.
2. In the target database, turn off the cable type dependency feature.
3. Change the wire color of the reference cables.
4. Turn on the cable type dependency feature.
5. In the Wiring module, click **Actions > Apply Regional Wire Colors**.
6. On the **Apply Regional Wire Colors** dialog box, click **Apply**.

NOTES

- You can view the results on the **Regional Wire Colors Application Results** dialog box.
- The software applies the wire colors only to those project cables that fully conform to the structure of reference cables on the basis of which they were built. That is, project cables must have the same number of cable sets and wires in each set as the appropriate reference cables.

SECTION 5

Conventional Panels

The Wiring module enables you to create and configure various types of conventional panels. Conventional panels in SmartPlant Instrumentation refer to all types of panels and cabinets that do not have a plug-and-socket connection. In **SmartPlant Instrumentation Explorer**, panels occupy the top hierarchy level under which you can create various wiring items such as terminal strips, I/O cards, racks, and so on. There is a lot of flexibility in creating a panel hierarchy. There is no rigid structure like "panel — strip — terminal" that limits your wiring design. You can create various wiring structures, as you require. See Panels by Category Hierarchy Example to examine a few possible structures.

Note that when duplicating a panel that has another panel as its child item (usually a device panel, the software duplicates the parent item without the child panel.

SmartPlant Instrumentation allows you to create the following types (categories) of conventional panels:

- Junction boxes
- Marshaling racks
- Cabinets (multi-purpose panels)
- Device panels
- Distributed Control System (DCS) panels
- Programmable Logic Controller (PLC) panels

Create a Junction Box

This procedure explains how to create a generic junction box.

You can create new panels either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that first you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the place where it will be physically located. This is helpful when filtering the panels in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder. For more information, see Create a Reference Item.
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder to display all the panel category folders.
 - Expand the **Panels by Location** hierarchy and right-click a location where you want your new panel to reside.
3. In the **Domain Explorer** under **Panels by Category** or in the **Reference Explorer** under **Panels**, right-click the **Junction Boxes** folder and then click **New > Junction Box (Generic)**.
4. In the **Junction Box Properties** dialog box, on the **General** tab, do one of the following to define the name of the new panel:

- Clear the **Apply naming convention** check box and under **Panel**, type the name of the new panel. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Panel**.
 - Keep the **Apply naming convention** check box selected if you want the software to name the new panel automatically according to the naming conventions that are set for this type of panel in the Administration module. For more information, see SmartPlant Instrumentation Administration Help, Domain Administration > Naming Conventions.
5. Select the required values from the **Type**, **Manufacturer**, **Model**, **Dimensions**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table. (For details, see Add a New Value to a Select List.)
 6. Under **Dimensions**, type a value for the marshaling rack dimensions if needed.
 7. Under **Maximum number of racks**, type a value to set the maximum number of racks that can be created in this panel.
TIP Setting this value to zero means that the current panel can have an unlimited number of racks.
 8. Under **Mounting**, type a value if needed.
 9. Under **Backplane**, type a value if needed.
 10. Select the **Set as intrinsically safe** check box if this junction box has intrinsic safety certification.
 11. Click the **Associate Symbols** tab to associate a symbol with the current panel. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
 12. Click **Revisions** to manage the revisions of the new panel if needed.
 13. Click **OK** to accept your definitions for the new junction box and close this dialog box.
NOTE To create a junction box that contains both terminals and plug-and-socket connections, see *Create a Panel with Terminals and Connectors* (on page 44).

Create a Marshaling Rack

This procedure explains how to create a marshaling rack.

You can create new panels either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that first you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the place where it will be physically located. This is helpful when filtering the panels in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder. For more information, see *Create a Reference Item*.
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder to display all the panel category folders.
 - Expand the **Panels by Location** hierarchy and right-click a location where you want your new panel to reside.

3. In the **Domain Explorer** under **Panels by Category** or in the **Reference Explorer** under **Panels**, right-click the **Marshaling Racks** folder and then click **New > Marshaling Rack**.
4. In the **Marshaling Rack Properties** dialog box, on the **General** tab, do one of the following to define the name of the new panel:
 - Clear the **Apply naming convention** check box and under **Panel**, type the name of the new panel. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Panel**.
 - Keep the **Apply naming convention** check box selected if you want the software to name the new panel automatically according to the naming conventions that are set for this type of panel in the Administration module. For more information, see SmartPlant Instrumentation Administration Help, Domain Administration > Naming Conventions.
5. Select the required values from the **Type**, **Manufacturer**, **Model**, **Dimensions**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table. (For details, see Add a New Value to a Select List.)
6. Under **Dimensions**, type a value for the marshaling rack dimensions if needed.
7. Under **Maximum number of racks**, type a value to set the maximum number of racks that can be created in this panel.

TIP Setting this value to zero means that the current panel can have an unlimited number of racks.
8. Under **Mounting**, type a value if needed.
9. Under **Backplane**, type a value if needed.
10. Select the **Set as intrinsically safe** check box if this marshaling rack has intrinsic safety certification.
11. If you need to enter power supply properties, select the **Requires power supply** check box and then click the **Power Supply** tab. For details, see *Enter Power Supply Data for Panels and Instrument Tags* (on page 45).
12. Click the **Associate Symbols** tab to associate a symbol with the current panel. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
13. Click **Revisions** to manage the revisions of the new panel if needed.
14. Click **OK** to accept your definitions for the new marshaling rack and close this dialog box.

NOTE To create a marshaling rack that contains both terminals and plug-and-socket connections, see *Create a Panel with Terminals and Connectors* (on page 44).

Create a Cabinet

This procedure explains how to create a cabinet.

You can create new panels either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that first you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the place where it will be physically located. This is helpful when filtering the panels in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.

- Press F8 to open the **Reference Explorer** and expand the **Panels** folder. For more information, see *Create a Reference Item*.
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder to display all the panel category folders.
 - Expand the **Panels by Location** hierarchy and right-click a location where you want your new panel to reside.
3. In the **Domain Explorer** under **Panels by Category** or in the **Reference Explorer** under **Panels**, right-click the **Cabinets** folder and then click **New > Cabinet**.
4. In the **Cabinet Properties** dialog box, on the **General** tab, do one of the following to define the name of the new panel:
 - Clear the **Apply naming convention** check box and under **Panel**, type the name of the new panel. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Panel**.
 - Keep the **Apply naming convention** check box selected if you want the software to name the new panel automatically according to the naming conventions that are set for this type of panel in the Administration module. For more information, see *SmartPlant Instrumentation Administration Help, Domain Administration > Naming Conventions*.
5. Select the required values from the **Type**, **Manufacturer**, **Model**, **Dimensions**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table. (For details, see *Add a New Value to a Select List*.)
6. Under **Dimensions**, type a value for the cabinet dimensions if needed.
7. Under **Maximum number of racks**, type a value to set the maximum number of racks that can be created in this panel.

TIP Setting this value to zero means that the current cabinet can have an unlimited number of racks.
8. Under **Mounting**, type a value if needed.
9. Under **Backplane**, type a value if needed.
10. Select the **Set as intrinsically safe** check box if this cabinet has intrinsic safety certification.
11. If you need to enter power supply properties, select the **Requires power supply** check box and then click the **Power Supply** tab. For details, see *Enter Power Supply Data for Panels and Instrument Tags* (on page 45).
12. Click the **Associate Symbols** tab to associate a symbol with the current panel. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
13. Click **Revisions** to manage the revisions of the new cabinet if needed.
14. Click **OK** to accept your definitions for the new cabinet and close this dialog box.

NOTE To create a cabinet that contains both terminals and plug-and-socket connections, see *Create a Panel with Terminals and Connectors* (on page 44).

Create an Unassociated Conventional Device Panel

This procedure explains how to create a device panel that is not associated with any tag number and not connected to any device cable.

You can create new panels either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that first you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the place where it will be physically located. This is helpful when filtering the panels in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder. For more information, see *Create a Reference Item*.
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder to display all the panel category folders.
 - Expand the **Panels by Location** hierarchy and right-click a location where you want your new panel to reside.
3. In the **Domain Explorer** under **Panels by Category** or in the **Reference Explorer** under **Panels**, right-click the **Device Panels** folder and then click **New > Device Panel (Conventional)**.
4. In the **Device Panel Properties** dialog box, on the **General** tab, do one of the following to define the name of the new panel:
 - Clear the **Apply naming convention** check box and under **Panel**, type the name of the new panel. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Panel**.
 - Keep the **Apply naming convention** check box selected if you want the software to name the new panel automatically according to the naming conventions that are set for this type of panel in the Administration module. For more information, see *SmartPlant Instrumentation Administration Help, Domain Administration > Naming Conventions*.
5. In the **Address 1** and **Address 2** fields, type the required definitions.

TIP You can change the **Address 1** and **Address 2** field headers to reflect manufacturer specific terminology. For details, see *Customize Manufacturer-Specific Field Headers*.
6. Select the required values from the **Type**, **Manufacturer**, **Model**, **Dimensions**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table. (For details, see *Add a New Value to a Drop-Down List*.)
7. Under **Dimensions**, type a value for the marshaling rack dimensions if needed.
8. Under **Mounting**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this marshaling rack has intrinsic safety certification.
10. Click the **Associate Symbols** tab to associate a symbol with the current panel. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new panel if needed.

12. Click **OK** to accept your definitions for the new device panel and close this dialog box.

NOTE To create a device panel that contains both terminals and plug-and-socket connections, see *Create a Panel with Terminals and Connectors* (on page 44).

Create a DCS (Distributed Control System) Panel

This procedure explains how to create a DCS (distributed control system) panel.

You can create new panels either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that first you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the place where it will be physically located. This is helpful when filtering the panels in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder. For more information, see *Create a Reference Item*.
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder to display all the panel category folders.
 - Expand the **Panels by Location** hierarchy and right-click a location where you want your new panel to reside.
3. In the **Domain Explorer** under **Panels by Category** or in the **Reference Explorer** under **Panels**, right-click the **DCS Panels** folder and then click **New > DCS Panel**.
4. In the **Distributed Control System (DCS) Properties** dialog box, on the **General** tab, do one of the following to define the name of the new panel:
 - Clear the **Apply naming convention** check box and under **Panel**, type the name of the new panel. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Panel**.
 - Keep the **Apply naming convention** check box selected if you want the software to name the new panel automatically according to the naming conventions that are set for this type of panel in the Administration module. For more information, see *SmartPlant Instrumentation Administration Help, Domain Administration > Naming Conventions*.
5. In the **Highway** and **Node** fields, type the required DCS definitions.

TIP You can change the **Highway** and **Node** field headers to reflect manufacturer-specific terminology. For details, see *Customize Manufacturer-Specific Field Headers*.
6. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table. (For details, see *Add a New Value to a Select List*.)
7. Under **Maximum number of racks**, type a value to set the maximum number of racks that can be created in this panel.

TIP Setting this value to zero means that the current panel can have an unlimited number of racks.
8. To create a DCS panel compatible with a Profibus DP segment, do the following:
9. Select the **Enable DP Profibus** check box.
10. Under **DP Profibus node**, type a node for the new panel.

11. If you need to enter power supply properties, select the **Requires power supply** check box and then click the **Power Supply** tab. For details, see *Enter Power Supply Data for Panels and Instrument Tags* (on page 45).
12. Click the **Associate Symbols** tab to associate a symbol with the current panel. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
13. Click **Revisions** to manage the revisions of the new panel if needed.
14. Click **OK** to accept your definitions for the new DCS panel and close this dialog box.

NOTES

- To create a DCS panel that contains both terminals and plug-and-socket connections, see *Create a Panel with Terminals and Connectors* (on page 44).
- Two detail symbols are supplied with the software, the **DcsDetail.sym** which displays the CS tags first and then the I/O and channel data, and the **Alt_DcsDetail.sym** which displays the CS tags below the I/O and channel data. You can replace the **DcsDetail.sym** with the **Alt_DcsDetail.sym** in two ways:
 - a. By changing the name of the **Alt_DcsDetail.sym** to **DcsDetail.sym**, this will replace the existing symbol with the alternative (this cannot be undone).
 - b. By selecting the **Alt_DcsDetail.sym** as the Symbol Detail File in the Panel Manufactures dialog box for a specific manufacture, this way panels associated with the specific manufacture will use the **Alt_DcsDetail.sym** instead of the **DcsDetail.sym**.

Create a PLC (Programmable Logic Controller) Panel

This procedure explains how to create a PLC (programmable logic controller) panel.

You can create new panels either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that first you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the place where it will be physically located. This is helpful when filtering the panels in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**. For more information, see *Create a Reference Item*.
2. Do one of the following:
 - Double-click the **Panels by Location** folder, right-click a location and then on the shortcut menu click **New > PLC Panel**.
 - Double-click the **Panels by Category** folder, right-click the **DCS Panels** folder and then on the shortcut menu click **New > PLC Panel**.
3. In the **Programmable Logic Controller (PLC) Properties** dialog box, on the **General** tab, do one of the following to define the name of the new panel:
 - Clear the **Apply naming convention** check box and under **Panel**, type the name of the new panel. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Panel**.

- Keep the **Apply naming convention** check box selected if you want the software to name the new panel automatically according to the naming conventions that are set for this type of panel in the Administration module. For more information, see SmartPlant Instrumentation Administration Help, *Domain Administration > Naming Conventions*.
- 4. In the **Address 1** and **Address 2** fields, type the required PLC definitions.
TIP You can change the **Address 1** and **Address 2** field headers to reflect manufacturer-specific terminology. For details, see *Customize Manufacturer-Specific Field Headers*.
- 5. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table. (For details, see *Add a New Value to a Select List*.)
- 6. Under **Maximum number of racks**, type a value to set the maximum number of racks that can be created in this panel.
TIP Setting this value to zero means that the current panel can have an unlimited number of racks.
- 7. To create a PLC panel compatible with a Profibus DP segment, do the following:
- 8. Select the **Enable DP Profibus** check box.
- 9. Under **DP Profibus node**, type a node for the new panel.
- 10. If you need to enter power supply properties, select the **Requires power supply** check box and then click the **Power Supply** tab. For details, see *Enter Power Supply Data for Panels and Instrument Tags* (on page 45).
- 11. Click the **Associate Symbols** tab to associate a symbol with the current panel. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
- 12. Click **Revisions** to manage the revisions of the new panel if needed.
- 13. Click **OK** to accept your definitions for the new PLC panel and close this dialog box.
NOTE To create a PLC panel that contains both terminals and plug-and-socket connections, see *Create a Panel with Terminals and Connectors* (on page 44).

Create a Panel with Terminals and Connectors

Use this procedure to create a panel that contains both terminals and plug-and-socket connections. Some fieldbus and Telecom devices have inputs via a plug-and-socket arrangement and outputs via terminated wires on terminals. To provide for this need, SmartPlant Instrumentation allows you to add a connector under any panel type (except for device panels) and then make the required connections either through the terminals or the connectors.

1. In the **Domain Explorer** or **Reference Explorer**, create or select a panel.
TIP You can select or create any type of panel except for device panels.
2. Right-click the panel you selected and then on the shortcut menu, click **New > Connector**.
3. In the **Connector Properties** dialog box, under **Connector**, type the name of the new connector.
4. Under **Connector type**, do one of the following:
 - From the list, select a connector type with pins.
 - To modify or create a connector type, click . For details, see *Making Connections and Associations for Fieldbus* (on page 199).
5. Under **Male/female** select **Male** or **Female** as needed.

6. Click **OK**.

Enter Power Supply Data for Panels and Instrument Tags

This procedure explains how to enter power supply data for various panels that require power supply. Although power supply data is not essential for SmartPlant Instrumentation functionality, it is indicated for the interface with the SmartPlant Electrical application. SmartPlant Instrumentation allows you to enter power supply data in all panels except for junction boxes and device panels.

1. Do one of the following to open the **Properties** dialog box of a panel or tag number:
 - In the **Domain Explorer** or **Reference Explorer**, right-click a desired panel and then click **Properties**.
 - In the **Domain Explorer** or **Reference Explorer**, right-click a desired instrument and then click **Properties**.
2. On the **General** tab, select the **Requires power supply** check box.
3. Click the **Power Supply** tab.
4. Under **Criticality**, select the criticality, which is used in SmartPlant Electrical to determine whether the current instrument requires UPS power or regular power.
5. Under **Rated voltage**, select a value that indicates the voltage at which the instrument is designed to operate.
6. If the instrument is running on alternating current, under **Power supply type** click **AC** and do the following:
 - a. Under **Frequency**, select the operating frequency of the instrument.
 - b. Under **Number of phases**, select the number of phases in the power supply to the instrument.
 - c. Under **Power factor full load**, enter a value between 0 and 1 for the ratio of active to apparent power at 100% of the rated power.
7. If the instrument runs on direct current, under **Power supply type**, click **DC**.
8. Under **Rated active load**, type the full load power consumption in kilowatts.
9. Type the following values in amperes:
 - a. Full load current.
 - b. Starting current.
10. In the **Coincidence factors** group box, do one of the following:
 - Under **Operating mode**, select **Continuous**, and under **X — continuous**, type a value between 0 and 1.
 - Under **Operating mode**, select **Intermittent**, and under **Y — intermittent**, type a value between 0 and 1.
 - Under **Operating mode**, select **Spare**, and under **ZZ — spare**, type a value between 0 and 1.
 - Under **Operating mode**, select **Standby**, and under **Z — stand by**, type a value between 0 and 1.

NOTE The fields under **Power distribution board data** group box display data that is determined within SmartPlant Electrical. On the **Power Supply** tab, click **Help** for more information.

Create a Rack

This procedure deals with the creation of racks. Since racks are panel child items, you can add a new rack under any panel except for device panels.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder. Then, select a desired reference panel. For more information, see *Create a Reference Item*.
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder to display all the panel category folders.
 - Expand the **Panels by Location** hierarchy to display the available locations and their panels. Then, select a desired panel.
3. In the **Domain Explorer** or in the **Reference Explorer**, right-click the selected panel and then click **New > Rack**.
4. In the **Rack Properties** dialog box, on the **General** tab, do one of the following to define the name of the new panel:
 - Clear the **Apply naming convention** check box and under **Rack**, type the name of the new rack. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Rack**.
 - Keep the **Apply naming convention** check box selected if you want the software to name the new panel automatically according to the naming conventions that are set for racks in the Administration module. For more information, see *SmartPlant Instrumentation Administration Help, Domain Administration > Naming Conventions*.
5. In the **Details** group box, type a description and sequence as you require.
6. Click **Slots** to add some slots for the current rack.
7. In the **Batch Slot Creation** dialog box, type the number of new slots that you want to add in the new rack.
8. Do the following to define the slot numbering:
9. Type a prefix for the slot name.
10. Enter a value in the **Start from number** and **Increment by fields**.
11. Click **OK** and to add the new slots return to the **Rack Properties** dialog box.

TIP In the **Rack Properties** dialog box, the software now displays the number of slots that you added.

12. Click the **Associate Symbols** tab to associate a symbol with the current rack. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).

NOTE When duplicating a rack that contains child items, the software duplicates the rack together with its child items.

Create a Slot

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder. Then, select a desired reference panel. For more information, see [Create a Reference Item](#).
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder to display all the panel category folders.
 - Expand the **Panels by Location** hierarchy to display the available locations and their panels. Then, select a desired panel.
3. In the **Domain Explorer** or in the **Reference Explorer**, expand the hierarchy of a panel to display the existing racks.
4. Right-click a rack and then click **New > Slot**.
5. On the **Slot Properties** dialog box, enter the values for the new slot as you require and click **OK**.

SECTION 6

Device Panels and Cables

This section describes the creation and management of device panels and device cables in SmartPlant Instrumentation.

In This Section

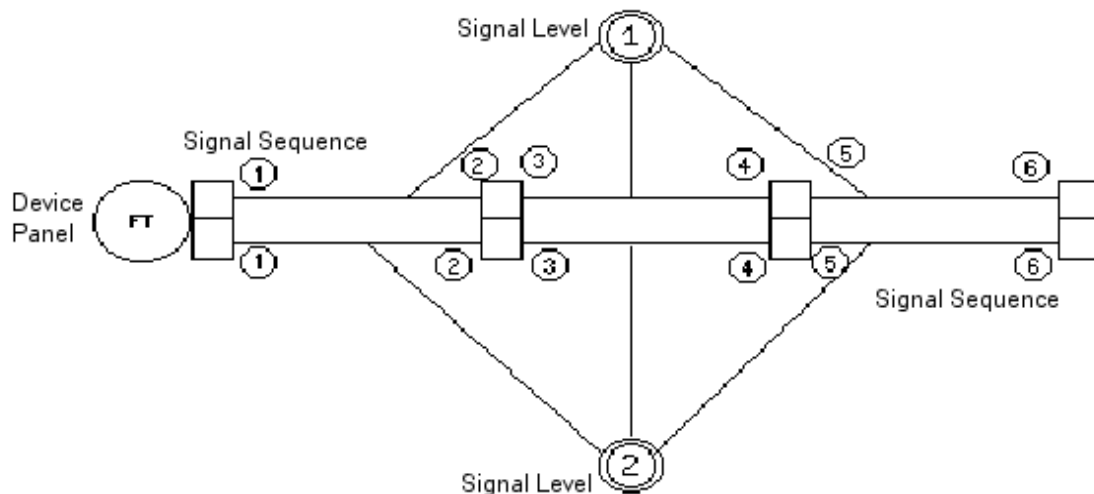
Device Panels in SmartPlant Instrumentation	49
Principles of Creating Device Panels	50
Create an Instrument with an Automatically Configured Device Panel	51
Create an Instrument with a Manually Configured Device Panel..	52
Create an Instrument Without a Device Panel	54
Create Device Panels and Cables in the Instrument Index Standard Browser View	55
Create Device Panels for Selected Instruments.....	55
Associate Instruments with Device Panels.....	57
Dissociate Device Panels from Instruments	57
Create a Device Cable.....	58
Rename Device Cables in Batch Mode.....	59

Device Panels in SmartPlant Instrumentation

A device panel in SmartPlant Instrumentation is a wiring item that has wiring and instrument tag information, for example transmitters, transducers, sensors, and so forth.

Wiring information consists of a signal (a wire group), signal level, and signal sequence. An instrument generates a signal which is carried through a group of wires. These wires are connected to a device panel associated with that instrument. The software propagates the signal through the wires on different levels. Signal levels can be, for example, plus, minus, and shield. A signal sequence is a method used by SmartPlant Instrumentation to assign numbers, in ascending or descending increments, to connection points along the signal propagation path at the same signal level.

The following diagram indicates how a device panel is associated with an instrument. You may then continue the wiring routing as required.



Each signal level defines a signal routing associated with a particular loop. It is possible to have up to 99 levels within one signal.

Note that SmartPlant Instrumentation automatically tracks signals and assigns the signal, signal level, and signal sequence values when you create a device panel.

Also, if a device panel is a child item of another panel, the software does not duplicate the child device panel when duplicating the parent panel.

Principles of Creating Device Panels

SmartPlant Instrumentation offers the following options for device panel creation:

- Creation of an individual device panel in the **Domain Explorer**. This device panel will not be associated with any instrument and will not have a device cable.
- Creation of a device panel for an existing instrument and configure its connection manually or automatically.
- Creation of a new instrument with an associated device panel and device cable. You can configure the connection automatically or manually as desired.

SmartPlant Instrumentation provides you with an easy to use mechanism to create device panels and their wiring directly from the Instrument Index module. Alternatively, you can create the connection information manually in the **Device Panels** window in the Wiring module, including the terminals, the cable, and the connections between them. The **Device Panels** window can help create the connection information needed in most cases where a single cable is connected to the terminals of the instrument.

Before creating a device panel with wiring information you need to define a number of reference items. These reference items are the panel, cable, and connection type. You can also define appropriate instrument type profiles that include wiring definitions. When creating an instrument based on a defined instrument type profile, the software automatically checks if the instrument that is being added needs to have a device panel. Based on the given default settings, the software will create the panel, the cable, and the connections between them. Also, SmartPlant Instrumentation automatically propagates the signals and names the device panels and cables according to the appropriate naming conventions defined in the Administration module. If no naming convention for device cables has been defined, the software names the cable after the instrument, when the name is prefixed with "C-".

Create an Instrument with an Automatically Configured Device Panel

This option enables you to create a new instrument with an associated device panel and device cable. In this case, SmartPlant Instrumentation automatically creates a device panel with a device cable and automatically configures the required connection. SmartPlant Instrumentation does not prompt you to select a reference panel, reference cable, or a connection type. The software selects the default configuration for the new device panel, device cable, and connection type that you have defined in the **Wiring and I/O Profile** group box of the **Instrument Type Profile** dialog box in the Instrument Index module. The new device panel will have the same name as the new instrument. SmartPlant Instrumentation will create the new instrument according to the wiring profile defined for it in the Instrument Index module. Make sure that you set and activate all the required instrument type profiles before you start creating new instruments.

1. Activate the instrument type profile default settings as follows:
 - a. Click **File > Preferences**.
 - b. In the tree view pane, to expand the tree, click  beside **Instrument Index**.
 - c. Click **Profile**.
 - d. Under **Activate instrument type profile options**, select the **Wiring** check box.
 - e. Click **OK**.
2. In the **Wiring Module** window, do one of the following:
 - Click .
 - On the menu bar click **Actions > Device Panels**.
3. In the **Device Panels** window, clear the **Manual configuration** check box.
4. Do one of the following:
 - Click .
 - On the menu bar click **Actions > New Tag**.
5. In the **New Tag Number** dialog box, do the following:
 - a. From the **Tag class** list, select the desired tag class according to whether you want to create a conventional instrument, Foundation Fieldbus instrument, HART instrument, Telecom instrument, and so forth.
 - b. Type the name of the new instrument. If needed, it is possible for instruments and device panels to have the same names. Make sure that you follow the naming conventions for new tag numbers. For details, see Instrument Tag Naming Convention.
 - c. Click **OK** to create the instrument.
 - d. If more than one function identifier exists for the current instrument type acronym, on the **Select Instrument Type** dialog box, select the appropriate instrument type, and then, click **OK**.

TIPS

- If the instrument name does not correspond to an existing loop name, SmartPlant Instrumentation prompts you to enter a loop name based on the tag number you have entered. The software automatically associates the new instrument with the loop if the loop identifier already matches an existing loop.

- If the loop convention includes the loop function parameter, the prompt always appears. In this case, you have to complete the loop number. If a profile exists for the selected instrument type, the software creates the new instruments for that instrument type according to the selected reference items. For details, see Define an Instrument Type Profile.
- 6. In the **Loop Name** dialog box, do one of the following:
 - Type the loop number name that the new instrument will be associated with.
 - Accept the displayed value to create a new loop with this name.
- 7. In the **Loop Name** dialog box, do one of the following:
 - Click **OK** to create the new loop number.
 - Click **Cancel** to create the new instrument without a loop association.
- 8. On the **Tag Number Properties** dialog box, edit the instrument properties as you require and click **OK**.

NOTE The software adds the new instrument to the list of instruments in the **Device Panels** window. Also, you can see the new instrument in the **Instruments** folder of the **Domain Explorer**.

Create an Instrument with a Manually Configured Device Panel

You use this option to create a new instrument with an associated device panel and device cable. In this case, SmartPlant Instrumentation creates a device panel and a device cable and lets you manually configure the required connection. The software prompts you to select a reference device panel and reference cable that will be used to create the new device panel and cable. SmartPlant Instrumentation copies the definitions for the new device panel and cable from the selected reference panel and cable that you defined in the **Reference Explorer**. You can also select an existing connection type and you can name the new device panel as you require. SmartPlant Instrumentation creates the new instruments according to the wiring profile that you defined in the Instrument Index module. Make sure that you set and activate all the required instrument type profiles before you start creating new instruments.

1. Activate the instrument type profile default settings as follows:
 - a. Click **File > Preferences**.
 - b. In the tree view pane, to expand the tree, click  beside **Instrument Index**.
 - c. Click **Profile**.
 - d. Under **Activate instrument type profile options**, select the **Wiring** check box.
 - e. Click **OK**.
2. In the **Wiring Module** window, do one of the following:
 - Click .
 - On the menu bar click **Actions > Device Panels**.
3. In the **Device Panels** window, select the **Manual configuration** check box.
4. Do one of the following:
 - Click .
 - On the menu bar click **Actions > New Tag**.
5. On the **New Tag Number** dialog box, do the following:

- a. From the **Tag class** list, select the desired tag class according to whether you want to create a conventional instrument, Foundation Fieldbus instrument, HART instrument, Telecom instrument, and so forth.
- b. Type the name of the new instrument. If needed, it is possible for instruments and device panels to have the same names. Make sure that you follow the naming conventions for new tag numbers. For details, see Instrument Tag Naming Convention.
- c. Click **OK** to create the instrument.
- d. If more than one function identifier exists for the current instrument type acronym, on the **Select Instrument Type** dialog box, select the appropriate instrument type, and then, click **OK**.

TIPS

- If the instrument name does not correspond to an existing loop name, SmartPlant Instrumentation prompts you to enter a loop name based on the tag number you have entered. The software automatically associates the new instrument with the loop if the loop identifier already matches an existing loop.
 - If the loop convention includes the loop function parameter, the prompt always appears. In this case, you have to complete the loop number. If a profile exists for the selected instrument type, the software creates the new instruments for that instrument type according to the selected reference items. For details, see Define an Instrument Type Profile.
6. In the **Loop Name** dialog box, do one of the following:
 - Type the loop number name that the new instrument will be associated with.
 - Accept the displayed value to create a new loop with this name.
 7. In the **Loop Name** dialog box, do one of the following:
 - Click **OK** to create the new loop number.
 - Click **Cancel** to create the new instrument without a loop association.
 8. On the **Create Device Panel and Cable** dialog box, under **Device panel**, accept or the displayed name of the device panel that will be created with the current instrument.
 9. From the **Reference device panel** list, select a reference device panel that you want to use as a basis for the new device panel.

TIPS

- The displayed reference panel is the one that you defined in the Instrument Index module, in the **Instrument Type Profile** dialog box, under **Wiring and I/O Profile**.
 - The properties of the reference panel that you select determine whether one or both of the **Conventional connections** and **Plug-and-socket connections** group boxes are enabled, and the values that are initially displayed.
10. In the active group boxes, to define the device cables, do the following as necessary:
 - To modify the properties of a selected device cable connection, click **Properties**, and in the dialog box that opens, change the values as necessary.
 - To add an additional device cable connection, click **New**, and in the dialog box that opens, enter the appropriate values.
 - To delete selected cable connections, click **Delete**.
 11. Click **OK** to complete the process.
 12. On the **Tag Number Properties** dialog box, edit the instrument properties as you require and click **OK**.

NOTE The new device panel appears next to the selected instrument in the **Device Panels** window. SmartPlant Instrumentation displays a cable icon  next to the new device cable indicating that a device cable is connected to it.

Create an Instrument Without a Device Panel

This option enables you to create a new instrument in the **Device Panels** window without creating a device panel or device cable. You can then select an existing device panel and associate it with the new instrument. You can also create a new device panel for the new instrument if needed. Note that SmartPlant Instrumentation creates the new instrument according to the wiring profile defined for it in the Instrument Index module. Make sure that you set all the required instrument type profiles before you start creating new instruments.

1. Deactivate the instrument type profile default settings as follows:
 - a. Click **File > Preferences**.
 - b. In the tree view pane, to expand the tree, click  beside **Instrument Index**.
 - c. Click **Profile**.
 - d. Under **Activate instrument type profile options**, clear the **Wiring** check box.
 - e. Click **OK**.
2. In the **Wiring Module** window, do one of the following:
 - Click .
 - On the menu bar click **Actions > Device Panels**.
3. In the **Device Panels** window, do one of the following:
 - Click .
 - On the menu bar click **Actions > New Tag**.
4. In the **New Tag Number** dialog box, do the following:
5. From the **Tag class** list, select the desired tag class according to whether you want to create a conventional instrument, Foundation Fieldbus instrument, HART instrument, Telecom instrument, and so forth.
6. Type the name of the new instrument. If needed, it is possible for instruments and device panels to have the same names. make sure that you follow the naming conventions for new tag numbers. For details, see Instrument Tag Naming Convention.
7. Click **OK** to create the instrument.
8. If more than one function identifier exists for the current instrument type acronym, on the **Select Instrument Type** dialog box, select the appropriate instrument type, and then, click **OK**.

TIPS

- If the instrument name does not correspond to an existing loop name, SmartPlant Instrumentation prompts you to enter a loop name based on the tag number you have entered. The software automatically associates the new instrument with the loop if the loop identifier already matches an existing loop
 - If the loop convention includes the loop function parameter, the prompt always appears. In this case, you have to complete the loop number. If a profile exists for the selected instrument type, the software creates the new instruments for that instrument type according to the selected reference items For details, see Define an Instrument Type Profile.
9. In the **Loop Name** dialog box, do one of the following:
 - Type the loop number name that the new instrument will be associated with.

- Accept the displayed value to create a new loop with this name.
10. In the **Loop Name** dialog box, do one of the following:
 - Click **OK** to create the new loop number.
 - Click **Cancel** to create the new instrument without a loop association.
 11. On the **Tag Number Properties** dialog box, edit the instrument properties as you require and click **OK**.

NOTE The software adds the new instrument to the list of instruments in the **Device Panels** window. Also, you can see the new instrument in the **Instruments** folder of the **Domain Explorer**.

Create Device Panels and Cables in the Instrument Index Standard Browser View

This procedure shows you how to create a device panel with a device cable in the Instrument Index standard browser.

1. In the **Instrument Index Module** window, do one of the following to open the Instrument Index Standard Browser view:
 - On the menu bar, click **Actions > Standard Browser View**.
 - On the toolbar, click .
2. Select the desired tag numbers.
3. Do one of the following:
 - Right-click, and on the shortcut menu, click **Apply Profile > Create Device Panel and Cable**.
 - On the **Actions** menu, click **Apply Profile > Create Device Panel and Cable**.
4. On the **Results** dialog box, view the propagation results, and then click **Close**.

NOTE To make this feature work, you must select the **Create wiring** check box and choose a reference device panel on the **Wiring and Control System** tab in the instrument type profile of the selected tag number before you attempt to create a new device panel. You can also select a reference device cable if you want SmartPlant Instrumentation to create device cables with the device panels. For more details, see Define an Instrument Type.

Create Device Panels for Selected Instruments

This option enables you to create device panels associated with selected instruments. This process also includes the creation of the required device cables that will be connected to the new device panels. You can let SmartPlant Instrumentation configure the connection for you or you can choose the manual configuration option that allows you to configure the connection:

- **Automatic configuration** — the software creates a device panel, a device cable, and automatically configures the required connection for you. SmartPlant Instrumentation does not prompt you to select a reference panel, reference cable, or a connection type. Instead, the software selects the reference configuration for the new device panel, device cable, and connection type that you have defined in the **Wiring and I/O Profile** group box of the **Instrument Type Profile** dialog box in the Instrument Index module. The new device panel will have the same name as the new instrument. Make sure that you set and activate all the required instrument type profiles before you start creating new instruments.

- **Manual configuration** — SmartPlant Instrumentation creates a device panel and a device cable and lets you manually configure the required connection. The software prompts you to select a reference panel and reference cable that will be used to create the new device panel and cable. SmartPlant Instrumentation copies the definitions for the new device panel and cable from the selected reference panel and cable that you defined in the **Reference Explorer**. You can also select an existing connection type and name the new device panel as needed.
1. In the **Wiring Module** window, do one of the following:
 - Click .
 - On the menu bar click **Actions > Device Panels**.
 2. In the **Device Panels** window, do one of the following:
 - Select the **Manual configuration** check box to define the device panel and cable manually.
 - Clear the **Manual configuration** check box to define the device panel and cable automatically.
 3. On the menu bar, click **Actions > New Tag** or click  on the toolbar.
 4. Select the instruments for which you want to create device panels and cables. You can use the Ctrl and Shift keys to select multiple instruments.
 5. On the menu bar click **Actions > Create**.

TIPS

- If you cleared the **Manual configuration** check box, SmartPlant Instrumentation creates and configures the device panel automatically. The creation process ends at this point and the new device panels appear next to the selected instruments in the **Device Panels** window. The software displays a cable icon  next to the new device panel indicating that a device cable is connected to it.
 - If you selected the **Manual configuration** check box, the creation process continues.
6. On the **Create Device Panel and Cable** dialog box, under **Device panel**, accept the displayed name of the device panel that will be created with the current instrument.
 7. From the **Reference device panel** list, select a reference device panel that you want to use as a basis for the new device panel.

TIPS

- The displayed reference panel is the one that you defined in the Instrument Index module, in the **Instrument Type Profile** dialog box, under **Wiring and I/O Profile**.
 - The properties of the reference panel that you select determine whether one or both of the **Conventional connections** and **Plug-and-socket connections** group boxes are enabled, and the values that are initially displayed.
8. In the active group boxes, to define the device cables, do the following as necessary:
 - To modify the properties of a selected device cable connection, click **Properties**, and in the dialog box that opens, change the values as necessary.
 - To add an additional device cable connection, click **New**, and in the dialog box that opens, enter the appropriate values.
 - To delete selected cable connections, click **Delete**.
 9. To apply the values that you set to the selected instruments with incomplete wiring profiles, select **Apply to all selected instruments with incomplete profile**.

NOTE You can also create device panels and cables for selected instruments in the Instrument Index Standard Browser view. For details, see *Create Device Panels and Cables in the Instrument Index Standard Browser View* (on page 55).

Associate Instruments with Device Panels

This option enables you to associate an instrument with a device panel. This feature is useful when, for instance, you create a number of device panels in the **Domain Explorer** by copying a reference item from the **Reference Explorer**. The device panels you create in the **Domain Explorer** are not associated with any instruments and they are not connected to device cables. A device panel must be associated with an instrument before you can create a device cable for it.

1. In the **Wiring Module** window, do one of the following:
 - Click .
 - On the menu bar click **Actions > Device Panels**.
2. In the **Device Panels** window, select the **Manual configuration** check box.
3. In the **Device Panels** window, select the instruments you require and then do one of the following:
 - Click .
 - On the menu bar, click **Actions > Associate**.
4. On the **Select Device Panels** dialog box, select the appropriate device panels.
5. Click **OK**.

NOTE The new device panel appears next to the selected instrument in the **Device Panels** window. You can now create a device cable for these panels.

Dissociate Device Panels from Instruments

This feature enables you to dissociate an instrument from a device panel. When carrying out this procedure, SmartPlant Instrumentation automatically disconnects single cables belonging to the dissociated device panels. Furthermore, the software renames the disconnected cables to '?-' plus <name>. For example, a cable named 'C-FT -2221' is renamed to '?-C-FT -2221'.

SmartPlant Instrumentation renames the cable to draw your attention to the fact that the cable is not connected to any device. In addition, the software renames the wires to SPARE throughout the signal run.

1. In the **Wiring Module** window, do one of the following:
 - Click .
 - On the menu bar click **Actions > Device Panels**.
2. In the **Device Panels** window, select the instruments you require and then do one of the following:
 - Click .
 - On the menu bar, click **Actions > Dissociate**.

Create a Device Cable

You use this option to create a new device cable and connect it to a selected device panel. This feature is useful when you created a device panel in the **Domain Explorer**, associated it with an instrument in the **Device Panels** window, but have not yet defined a device cable for it. You can configure the cable and its connection type manually or you can let SmartPlant Instrumentation select the reference cable and connection type automatically.

1. In the **Wiring Module** window, do one of the following:
 - Click .
 - On the menu bar click **Actions > Device Panels**.
2. In the **Device Panels** window, do one of the following:
 - Select the **Manual configuration** check box to define the device panel and cable manually.
 - Clear the **Manual configuration** check box to define the device panel and cable automatically.
3. Select the required device panel – instrument associations for which you want to create device cables.
4. On the menu bar, click **Actions > Create**.

TIPS

- If you cleared the **Manual configuration** check box, SmartPlant Instrumentation creates and configures the device panel automatically. The creation process ends at this point and the new device panels appear next to the selected instruments in the **Device Panels** window. The software displays a cable icon  next to the new device panel indicating that a device cable is connected to it.
 - If you selected the **Manual configuration** check box, the creation process continues.
5. On the **Create Device Panel and Cable** dialog box, in the active group box (**Conventional connections** or **Plug-and-socket connections**) do the following for each device panel associated with an instrument:
 - a. Click **New**.
 - b. On the dialog box that opens, enter the appropriate values.
 - c. Click **OK**.
 6. On the **Create Device Panel and Cable** dialog box, click **OK**.

NOTE The new device cable appears in the **Device Panels** window next to the selected instrument.

Rename Device Cables in Batch Mode

This option enables you to select a panel with device cables (cables connecting a device panel and any other panel) and rename all the device panel cables connected to this panel in batch mode.

1. In the **Domain Explorer**, expand the **Panels by Location** or **Panels by Category** folder.
2. Right-click a junction box in the tree view or select several junction boxes in the **Items** pane and then right-click them.
3. On the shortcut menu, click **Actions > Rename Device Cables**.
4. On the **Rename** tab of the **Rename Device Cables** dialog box, select the check boxes next to the cables that you want to rename or choose the **Select All** check box to select all the cables displayed in the data window.
5. Click the **Convention** tab to define the cable name template, that is, the rules how the cable names will change.

TIP If a name template has been defined, it is displayed in the **Current Convention** field.

6. Insert and define as many conventions as required.
7. Select the **Use as default** check box to use the convention row definitions as your default naming conventions.
8. Click **Apply** when done.

NOTE You can modify the device panel name according to your needs. Device cable names are, however, not editable in this dialog box. You can rename device cables in the **Domain Explorer**.

SECTION 7

Wiring Equipment and Apparatus

This section describes the creation and management of wiring equipment and apparatuses.

In This Section

Wiring Equipment	61
Customize and Use Wiring Equipment Categories	62
Add User-Defined Wiring Equipment Category Properties	63
Create a Wiring Equipment Item	64
Wiring Items Hierarchy Examples	65
Create an I/O Card	68
Create an I/O Termination	70
Customize the Controller/Processor List Header in I/O Card Properties	71
Associate an I/O card with an I/O Termination	72
Apparatus.....	73
Define an Apparatus Configuration	73
Create an Apparatus Group.....	75
Add an Apparatus to an Apparatus Group	76
Edit Apparatus Terminal Properties.....	76
The Plug-and-Socket Group Wizard.....	77
Modify the Internal Links of a Plug-and-Socket Group	79

Wiring Equipment

SmartPlant Instrumentation enables you to create and manage various types of wiring equipment. Wiring equipment is a generic name for different termination objects that can handle multiple input and output signals (channels). These items need an external power supply and they usually have several terminals to feed the electronic components. Examples of wiring equipment are barriers, isolators, trip amplifiers, relays, and so forth. I/O cards, I/O termination items, and various terminal strips also belong to the group called wiring equipment.

Prior to Version 7, SmartPlant Instrumentation provided for this type of objects only by letting you create an apparatus. However, apparatus did not deal with certain aspects of wiring equipment such as electric properties. Apparatus provided a "flat" structure that could only group terminals on an apparatus strip under a position name. Wiring equipment, however, introduces a structured object which you can remove, add, and copy. You can accurately define wiring equipment properties such as category, manufacturer, model, and so forth.

NOTE When copying wiring equipment that has associated with it a plug-and-socket group, the Wiring preferences > Copy Items > Duplicate jumpers must be selected for the internal links to be copied along with the plug-and-socket group.

Wiring equipment is a much wider feature which allows you to create various types of termination objects and define their electric properties. You can use wiring equipment whenever you need to define any type of card, or piece of hardware that you need to buy separately, count the number of units you need (BOM), install it inside a panel, and make connections

SmartPlant Instrumentation enables you to create and manage wiring equipment items that:

- Contain various terminal strips or combination of terminal strips.
- Serve multiple tags or channels.
- Contain terminal strips and connectors.

- Require custom properties for each wiring equipment category.
- Require a specific presentation using special custom symbols.

Wiring equipment items are classified according to various categories. SmartPlant Instrumentation is shipped with a number of these categories. You can customize your own wiring equipment categories and create various wiring equipment items belonging to these customized categories. For more information, see the following:

- *Customize and Use Wiring Equipment Categories* (on page 62)
- *Add User-Defined Wiring Equipment Category Properties* (on page 63)

When designing a Foundation Fieldbus or Profibus system, you can create a wiring equipment item belonging to the fieldbus brick category. Then, under a fieldbus brick you can add a fieldbus apparatus. This compound object provides for the creation and management of fieldbus termination objects required for your Foundation Fieldbus or Profibus system. For details, see *Fieldbus Bricks* (on page 194).

Also, note the following features of wiring equipment:

- Wiring equipment can encapsulate multiple objects, terminal strips, and channels.
- Since every wiring equipment is a separate item, the **Connection** window cannot display multiple objects as it does for an apparatus strip. Nor is it possible to connect a cable to multiple wiring equipment items in one step.
- Using wiring equipment in combination with custom symbols provides for a better representation of loop drawings.
- Wiring equipment information does not appear in loop drawing generation. In most cases, it is possible to add macros and retrieve information or use custom symbols.
- Wiring reports do not show wiring equipment. All reports continue to show panels, terminal strips, and terminals.
- Wiring equipment can be displayed in cable block diagrams.
- Wiring equipment can be placed under other wiring equipment.

Customize and Use Wiring Equipment Categories

SmartPlant Instrumentation provides you with a number of predefined wiring equipment categories. However, you can also customize your own wiring equipment categories for wiring items that do not belong to any existing categories.

This procedure explains how to customize a wiring equipment category and add a customized property to a category. Note that some of the shipped wiring equipment categories have predefined properties. You can add your own customized properties to the shipped categories as well as to the customized ones.

After customizing a new wiring equipment category, it becomes available in the **Categories** list of the **New Wiring Equipment** dialog box and you can create new wiring equipment items belonging to this category.

1. On the Wiring module menu bar, click **Tables > Wiring Equipment > Categories**.
2. In the **Wiring Equipment Category** dialog box, click **New**.
3. In the **New Wiring Equipment Category** dialog box, type a category name and description.
4. To select a user-defined icon for the current wiring equipment category, under **Icon file name**, click **Browse** to navigate to the icon file that you require.

TIP The icon that you select will appear in the **Domain Explorer** or the **Reference Explorer** next to each wiring equipment item belonging to the current category.

5. In the **Properties** group box, click **Add**.
6. Under **Header**, type the text that will appear as the property name on the **Category Properties** tab of the appropriate wiring equipment properties dialog box.

7. From the **Type** list, select the type of value for the new customized category property. You will then be able to enter a value for this new property on the **Category Properties** tab of the appropriate wiring equipment properties dialog box. The values can be numeric, text, or date/time.
 8. Do one of the following:
 - Click **Add** to define another new property.
 - Click **OK** to accept your definitions and close this dialog box.
 9. Create a new wiring equipment item .

TIP In the **Domain Explorer**, right-click a panel or a panel child item, and then on the shortcut menu, click **New > Wiring Equipment**.
 10. In the **New Wiring Equipment** dialog box, from the **Category** list, select the newly created category, type the wiring equipment item name, and click **OK**.
 11. On the **General** tab of the **Properties** dialog box, define wiring equipment properties as you require and click the **Category Properties** tab.
 12. On the **Category Properties** tab, enter the values for each property as you require,
 13. Continue defining the properties of the new wiring equipment item as you need.
- NOTE** The software does not allow you to delete or rename the shipped wiring equipment categories. Nor can you delete or rename any of the predefined properties belonging to the shipped wiring equipment categories.

Add User-Defined Wiring Equipment Category Properties

This feature allows you to add user-defined properties to existing wiring equipment categories. This option applies to the user-defined categories as well as to those that have been supplied with SmartPlant Instrumentation.

1. On the Wiring module menu bar, click **Tables > Wiring Equipment > Categories**.
2. In the **Wiring Equipment Category** dialog box, select a category and click **Properties**.
3. To select a user-defined icon for the current wiring equipment category, under **Icon file name**, click **Browse** to navigate to the icon file that you require.

TIP The icon that you select will appear in the **Domain Explorer** or the **Reference Explorer** next to each wiring equipment item belonging to the current category.
4. In the **Wiring Equipment Category Properties** dialog box click **Add**.
5. Under **Header**, type the text that will appear as the property name on the **Category Properties** tab of the appropriate wiring equipment properties dialog box.
6. From the **Type** list, select the type of value for the new customized category property. You will then be able to enter a value for this new property on the **Category Properties** tab of the appropriate wiring equipment properties dialog box. The values can be numeric, text, or date/time.
7. Do one of the following:
 - Click **Add** to define another new property.
 - Click **OK** to accept your definitions and close this dialog box.

NOTES

- You cannot delete or rename any of the predefined properties belonging to the shipped wiring equipment categories.
- After adding a category property, it becomes available on the **Category Properties** tab of the appropriate wiring equipment properties dialog box.

Create a Wiring Equipment Item

This procedure explains how to create a wiring equipment item that belongs to any wiring equipment category except for I/O cards and I/O terminations.

You can create new wiring equipment items either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a wiring equipment item that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference wiring equipment items facilitate fast creation of plant items by copying an existing wiring equipment item from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a wiring equipment item that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the wiring equipment items in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder.
 2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder and navigate to a wiring item under which you want to create a new wiring equipment item .
 - Expand the **Panels by Location** folder and navigate to a wiring item under which you want to create a new wiring equipment item .
- TIP** There is a lot of flexibility in creating a wiring hierarchy in SmartPlant Instrumentation. There is no rigid structure like "panel – terminal strip – terminal" that limits your wiring design. Therefore, you can create wiring equipment items under various types of items. To see examples of possible wiring hierarchy structures, see *Hierarchy Examples of Wiring Items* (see "Wiring Items Hierarchy Examples" on page 65).
3. Right-click a wiring item under which you want to create new wiring equipment and then on the shortcut menu, click **New > Wiring Equipment**.
 4. In the **New Wiring Equipment** dialog box, from the **Category** list, select a category to which the new item belongs.
 5. Under **Name**, type the name of the item and click **OK**.
 6. In the **Wiring Equipment Properties** dialog box, on the **General** tab, do one of the following to define the name of the new equipment:
 - Select the **Apply naming convention** check box selected if you want the software to name the new equipment automatically according to the naming conventions that are set for this type of equipment in the Administration module. For more information, see SmartPlant Instrumentation Administration Help, Domain Administration > *Naming Conventions*.
 - Clear the **Apply naming convention** check box and under **Name**, type the name of the new equipment. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Name**.
 7. Select the **Double width** check box if the item occupies a double width slot.
 8. In the **Details** group box do the following as you require:
 - a. Type a description.
 - b. Select an item type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.

- c. Enter a sequence if you need to define the sequence of the new item in its parent item . This sequence also determines the order in which wiring items appear under their parent item in the **Domain Explorer** tree view.
9. Click the **Category Properties** tab.

TIP If you are creating a miscellaneous wiring equipment item , the **Category Properties** tab is not available.
10. Revise and modify category property values as you require. Click the value for each property and modify it as needed.

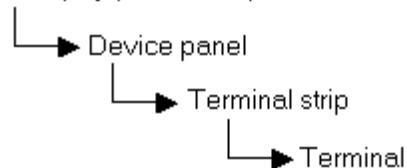
TIP Wiring equipment categories that are shipped with SmartPlant Instrumentation have predefined properties. You cannot delete or rename any of these categories or their properties. You can only edit their values. However, you can add user-defined properties to any category which you can rename or delete as you wish. For details, see *Customize and Use Wiring Equipment Categories* (on page 62).
11. Click the **Associate Symbols** tab to associate a symbol with the current item . The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
12. Click **OK** to accept your settings and close the dialog box.

Wiring Items Hierarchy Examples

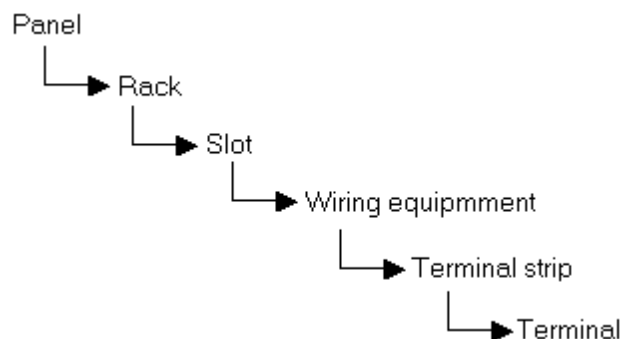
There is a lot of flexibility in creating a wiring hierarchy in SmartPlant Instrumentation. There is no rigid structure like "panel - terminal strip - terminal" that limits your wiring design. Therefore, you can create wiring equipment items under various types of items.

Example 1:

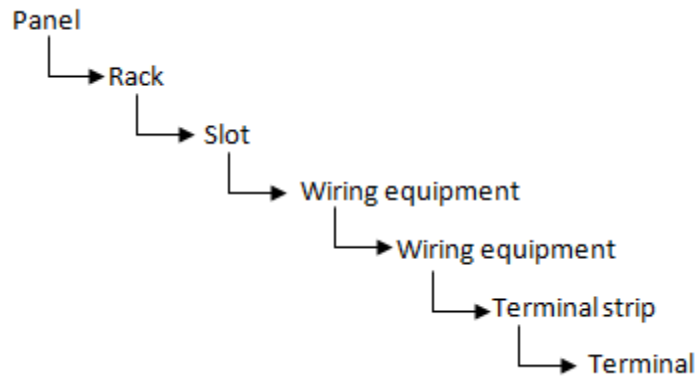
Panel (any panel except for a device panel)



Example 2:

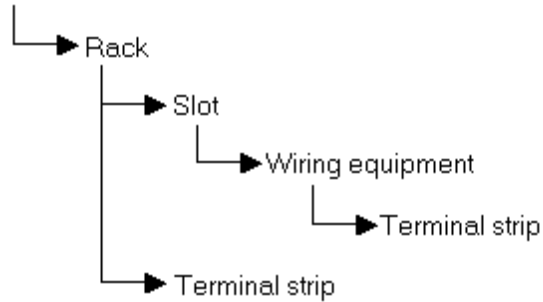


Example 3:

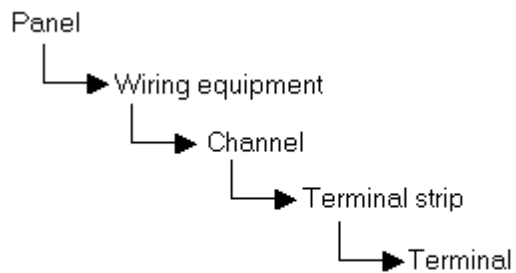


Example 4:

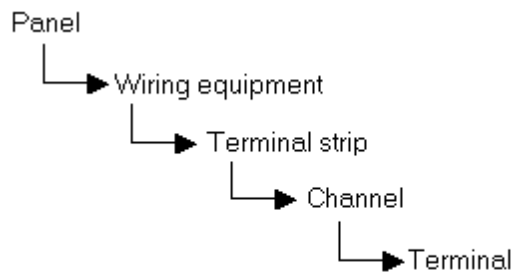
Panel (any panel except for a device panel)



Example 5:

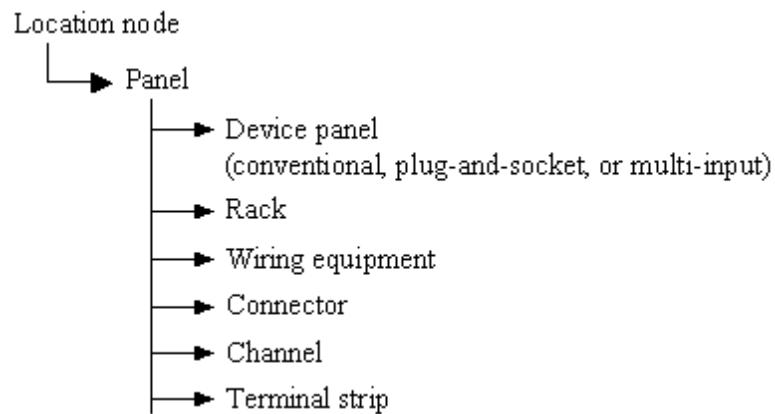


Example 6:



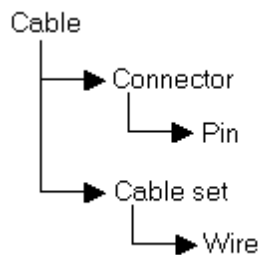
Example 7:

This example shows the wiring items that can exist in the **Panels by Location** folder.



Example 8:

This example shows the items that can exist in the **Cables** folder.



For a better understanding of the Domain Explorer, see [Working with the Domain Explorer](#).

Create an I/O Card

As a rule, I/O cards reside in slots. Therefore, you have to navigate to a particular rack and select a slot where you want to create a new I/O card. Note that I/O cards may or may not have built-in termination blocks. If an I/O card does not have a built-in termination block, you need to associate it with an I/O termination. To define an I/O card that has a built-in termination block, create an I/O card and then add a terminal strip with channels under this I/O card. For an I/O card with a built-in terminal block, you need to create a terminal strip with channels under an I/O card to be able to effect I/O assignment and make connections.

You can create new I/O cards either in the **Domain Explorer** or the **Reference Explorer**. If you are creating an I/O card that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference I/O cards facilitate fast creation of plant I/O cards by copying an existing item from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an I/O card that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering I/O cards in the current <unit>.

1. Press F7 to open the **Domain Explorer**.
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder and navigate to a wiring item under which you want to create a new I/O card.
 - Expand the **Panels by Location** folder and navigate to a wiring item under which you want to create a new I/O card.
3. Double-click a panel and then a rack till you see the existing slots.

TIP As a rule, I/O cards reside in rack slots. Therefore, you have to navigate to a slot where you want to create an I/O card. However, SmartPlant Instrumentation allows the creation of flexible wiring hierarchies and therefore you can add I/O cards to a number of various item types. To see examples of possible wiring hierarchy structures, see *Hierarchy Examples of Wiring Items* (see "Wiring Items Hierarchy Examples" on page 65).
4. Right-click a slot or any other wiring item under which you want the new card reside and then on the shortcut menu, click **New > Wiring Equipment**.
5. In the **New Wiring Equipment** dialog box, from the **Category** list, select **I/O Card**.
6. Under **Name**, type the name of the new I/O card and click **OK**.
7. In the **Wiring Equipment Properties – I/O Card** dialog box, on the **General** tab, do one of the following to define the name of the new card:
 - Select the **Apply naming convention** check box selected if you want the software to name the new card automatically according to the naming conventions that are set for this type of equipment in the Administration module. For more information, see SmartPlant Instrumentation Administration Help, *Domain Administration > Naming Conventions*.
 - Clear the **Apply naming convention** check box and under **Name**, type the name of the new card. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Name**.
8. Select the **Double width** check box if the card occupies a double width slot.
9. In the **Details** group box do the following as you require:
 - a. Type a description.
 - b. Select an I/O card type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.

- c. Enter a sequence if you need to define the sequence of the new card in its parent item . This sequence also determines the order in which wiring items appear under their parent item in the **Domain Explorer** tree view.
10. Click the **Control System** tab.
11. In the **Control system details** group box, under **I/O type**, select a I/O type to determine the actual function of the new I/O card. For example, **AI**, **AO**, **DI**, **DP**, **Fieldbus**, **Profibus**, and so forth. You can add more values to this list in the **I/O Type** supporting table in the Instrument Index module. (See the important notes at the end of this procedure.)
12. Under **Module**, type the software address that this card is assigned to.

TIP **Module**, **Controller/ Processor**, **Rack**, and **Slot** are field headers specific to panel manufacturers. Selecting a different manufacturer when editing the panel displays the headings used by that panel manufacturer. For details, see *Customize the Controller/Processor List Header in I/O Card Properties* (on page 71).
13. From the **Controller/Processor** list, select an I/O card controller. If the required controller is not available on the list, click  to define a new one.
14. To view the list of all I/O terminations associated with the current I/O card, click the **I/O Terminations** button.
15. Click the **Category Properties** tab.
16. Revise and modify category property values as you require. Click the value for each property and modify it as needed.

TIP **I/O card** is a wiring equipment category that is shipped with SmartPlant Instrumentation. You cannot delete or rename any of the wiring equipment categories that have been shipped with the software. Nor can you delete or rename the properties of the shipped categories. However, when editing I/O card properties, you can add user-defined properties which you can rename or delete as you wish. For details, see *Customize and Use Wiring Equipment Categories* (on page 62).
17. Click the **Associate Symbols** tab to associate a symbol with the current I/O cards. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
18. Click **OK** to accept your settings and close the dialog box.

IMPORTANT

- In a redundant system configuration, an I/O card can function as a primary or secondary I/O card and can be associated with several I/O terminations. Therefore, an I/O termination can serve multi-paired redundant I/O cards. However, in these redundant I/O card configurations, the I/O termination can be associated only with one pair of primary and secondary I/O cards. Also, an I/O card cannot function as both a primary and secondary I/O card for the same I/O termination.
- If an I/O card functions as a primary I/O card, the **Control System** tab of the **Wiring Equipment Properties – I/O Card** dialog box displays the secondary I/O card data and the primary I/O card data properties remain blank. If an I/O card functions as a secondary I/O card, the **Control System** tab displays the primary I/O card data and the secondary I/O card data properties remain blank.

Create an I/O Termination

I/O terminations are wiring equipment items that are used to connect I/O cards and other wiring items that do not have built-in termination blocks. You associate an I/O termination with an I/O card that does not have a built-in termination block and then make the required connections to the I/O termination. You can also associate an I/O termination with several I/O cards if there are redundant I/O cards.

Also, you can create an I/O termination when you need to define a distant I/O card that resides in a different panel or rack. You need to create a terminal strip with channels under an I/O termination to be able to effect I/O assignment and make connections.

You can create new I/O termination cards either in the **Domain Explorer** or the **Reference Explorer**. If you are creating an item that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference items facilitate fast creation of plant items by copying an existing item from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an I/O termination that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the wiring equipment items in the current <unit>.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder.
2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder and navigate to a wiring item under which you want to create a new I/O termination.
 - Expand the **Panels by Location** folder and navigate to a wiring item under which you want to create a new I/O termination.
3. Double-click a panel and then a rack till you see the existing slots.

TIP As a rule, I/O terminations reside in DCS panels or marshaling racks. However, SmartPlant Instrumentation allows to create flexible wiring hierarchies and therefore you can add I/O terminations to a number of various item types. To see examples of possible wiring hierarchy structures, see *Hierarchy Examples of Wiring Items* (see "Wiring Items Hierarchy Examples" on page 65).
4. Right-click a slot or any other wiring item under which you want the new card reside and then on the shortcut menu, click **New > Wiring Equipment**.
5. In the **New Wiring Equipment** dialog box, from the **Category** list, select **I/O Termination**.
6. Under **Name**, type the name of the new I/O termination and click **OK**.
7. In the **Wiring Equipment Properties – I/O Termination** dialog box, on the **General** tab, do one of the following to define the name of the new card:
 - Select the **Apply naming convention** check box selected if you want the software to name the new card automatically according to the naming conventions that are set for this type of equipment in the Administration module. For more information, see SmartPlant Instrumentation Administration Help, Domain Administration > Naming Conventions.
 - Clear the **Apply naming convention** check box and under **Name**, type the name of the new card. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Name**.
8. Select the **Double width** check box if the card occupies a double width slot.
9. In the **Details** group box do the following as you require:
 - a. Type a description.

- b. Select an I/O termination type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.
 - c. Enter a sequence if you need to define the sequence of the new I/O termination in its parent item. This sequence also determines the order in which wiring items appear under their parent item in the **Domain Explorer** tree view.
10. Click the **Control System** tab.
11. From the **I/O type** list, select a I/O type to determine the actual function of the new I/O termination. For example, **AI**, **AO**, **DI**, **DP**, **Fieldbus**, **Profibus**, and so forth. You can add more values to this list in the **I/O Type** supporting table in the Instrument Index module.
12. To associate the current I/O termination with an I/O card that does not have a built-in termination block, select the required options in the **Primary I/O card** group box.

TIP The **I/O card** list contains only those I/O cards that have been created under a rack and a slot. For more information, see *Associate an I/O card with an I/O Termination* (on page 72).
13. To define the current I/O termination as an I/O termination that serves two redundant I/O cards, do the following:
 - a. Select the **Define a redundant I/O card** check box.
 - b. Select the required options in the **Secondary I/O card** group box.
14. Click the **Category Properties** tab.
15. Revise and modify category property values as you require. Click the value for each property and modify it as needed.

TIP **I/O termination** is a wiring equipment category that is shipped with SmartPlant Instrumentation. You cannot delete or rename any of the wiring equipment categories that have been shipped with the software. Nor can you delete or rename the properties of the shipped categories. However, when editing I/O termination properties, you can add user-defined properties which you can rename or delete as you wish. For details, see *Customize and Use Wiring Equipment Categories* (on page 62).
16. Click the **Associate Symbols** tab to associate a symbol with the current I/O termination. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
17. Click **OK** to accept your settings and close the dialog box.

Customize the Controller/Processor List Header in I/O Card Properties

This option enables you to make the **Controller/Processor** list header on the **Control System** tab of the **I/O Card Properties** dialog box dependant on the panel manufacture to which the current I/O card belongs.

1. In the **Domain Explorer**, navigate to a panel.
2. Right-click the selected panel and then on the shortcut menu, click **Properties**.
3. On the **General** tab of the **Properties** dialog box, click  beside the **Manufacturer** list.
4. In the **Panel Manufacturers** supporting table, select a manufacturer and then scroll to the right to the **Controller/Processor** header.
5. Click in the **Controller/Processor** field and type the text that you want to appear instead of the **Controller/Processor** list header on the **Control System** tab of the **I/O Card Properties** dialog box.
6. Click **OK** to close the **Panel Manufacturers** supporting table.
7. Click **OK** in the **Panel Properties** dialog box.

Associate an I/O card with an I/O Termination

I/O terminations function as termination blocks for I/O cards that do not have built-in termination blocks. Therefore, you need to associate an I/O termination with an I/O card that needs a built-in termination block. You can associate only those I/O cards that have been created in a particular rack and slot.

IMPORTANT

- In a redundant system configuration, an I/O card can function as a primary or secondary I/O card and can be associated with several I/O terminations. Therefore, an I/O termination can serve multi-paired redundant I/O cards. However, in these redundant I/O card configurations, the I/O termination can be associated only with one pair of primary and secondary I/O cards. Also, an I/O card cannot function as both a primary and secondary I/O card for the same I/O termination.
 - If an I/O card functions as a primary I/O card, the **Control System** tab of the **Wiring Equipment Properties – I/O Card** dialog box displays the secondary I/O card data and the primary I/O card data properties remain blank. If an I/O card functions as a secondary I/O card, the **Control System** tab displays the primary I/O card data and the secondary I/O card data properties remain blank.
1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer** and expand the **Panels** folder.
 2. In the **Domain Explorer**, do one of the following:
 - Expand the **Panels by Category** folder, hierarchy and navigate to a wiring item under which you want to create a new I/O card.
 - Expand the **Panels by Location** hierarchy and navigate to a wiring item under which you want to create a new I/O card.
 3. Create or select an I/O card under a particular panel, rack, and slot.
 4. Create or select an I/O termination that you want to associate with an I/O card.
 5. In the **Wiring Equipment Properties – I/O Termination** dialog box, click the **Control System** tab.

TIP To open the **Wiring Equipment Properties – I/O Termination** dialog box, in the **Domain Explorer**, right-click an I/O termination and then on the shortcut menu, click **Properties**.

6. To associate a primary I/O card, in the **Primary I/O card** group box, click  next to the **I/O card** select list arrow.
7. In the **Domain Explorer**, select an appropriate I/O card and click **OK**.

TIPS

- You cannot select an I/O card whose child item is a terminal strip with channels.
 - Note that the I/O type of both the I/O card and the I/O termination must be the same. If the I/O type of the I/O termination differs from the I/O type of the I/O card, the software displays a warning.
8. To associate a secondary I/O card, do the following:
 - a. Click the **Define a redundant I/O card** check box.
 - b. In the **Secondary I/O card** group box, click  next to the **I/O card** select list arrow.
 9. Click **OK**.

NOTE To be able to effect I/O assignment and make connections, create a terminal strip with channels under the I/O termination you are editing.

Apparatus

Apparatus is a wiring item that pertains to different types of multi-purpose I/O devices. These devices accept wires and signals, for example, barriers, opto-couplers, relays, repeaters, isolators, and so forth. You also use this feature to create a fieldbus termination block whose configuration is different from an apparatus configuration.

SmartPlant Instrumentation allows you to create a group of apparatuses or a single apparatus. An apparatus group constitutes an apparatus strip (that is, a terminal strip) with a number of apparatuses created according to a selected apparatus configuration.

Adding a new apparatus group to a panel or a wiring equipment item starts with configuring the new apparatuses: setting the number of apparatuses on an apparatus strip, defining the apparatus terminal numbering pattern, signal propagation, and the color of the terminals. After defining an apparatus configuration, you can proceed with the creation of an apparatus group. The software opens the **Terminal Strip Properties** dialog box so that you can define an apparatus strip for the new apparatus group. After defining the apparatus strip properties in the **Terminal Strip Properties** dialog box, the software creates the apparatus strip and the apparatuses according the selected configuration.

SmartPlant Instrumentation also allows you to add a single apparatus to an apparatus group. In this case, you do not define an apparatus configuration. Then, you can select the new apparatus and add the required terminals.

Once you have created a new apparatus group or a single apparatus, SmartPlant Instrumentation displays it in the **Domain Explorer** showing the terminals belonging to the apparatus.

Note that an apparatus is a wiring item. Therefore, you can define apparatus properties, duplicate, or delete an apparatus, and assign it a special symbol. The software uses this symbol in the Enhanced Report Utility.

Define an Apparatus Configuration

You need to create an apparatus configuration before you can add an apparatus group to a selected panel or wiring equipment item. An apparatus group constitutes an apparatus strip (that is, a terminal strip) with a number of apparatuses created according to a selected apparatus configuration.

When defining an apparatus configuration, you define the apparatus profile (name, description, manufacturer, and model), the number of apparatuses, the apparatus terminal configuration, and the position numbering.

IMPORTANT To facilitate fast creation of apparatus groups with various configurations, we recommend that you create several apparatus configurations in the **Reference Explorer** so that you can create different reference apparatus groups that can be copied to the **Domain Explorer**.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Do one of the following:
 - In the **Domain Explorer**, double-click the **Panels by Location** folder, and select a panel.
 - In the **Domain Explorer**, double-click the **Panels by Category** folder, and select a panel.
 - In the **Reference Explorer**, double-click the **Panels** folder, and select a panel.
3. Right-click a panel to which you want to add an apparatus and then on the shortcut menu, click **New > Apparatus Group**.

4. In the **Apparatus** dialog box, click **New**.
5. In the **Apparatus profile** group box, in the **Configuration name** field, type the name of the new apparatus configuration.
6. Select the **Fieldbus apparatus** check box if you need to create a configuration for a fieldbus brick. Note that the apparatus configuration for a fieldbus brick differs from the conventional one. For details, see *Configure a Fieldbus Apparatus* (on page 197).
7. From the **Manufacturer** list, select the appropriate apparatus manufacturer. If the required value is not available, click  next to the list arrow to add or edit values for this list.
8. In the **Apparatus numbering** group box, type the position name in the **Prefix** field if required.
TIP Position names can be 15-character long. This name will appear in the **Domain Explorer** within the new strip (only the first four characters), in the **Connection** window, and in the appropriate reports.
9. Select the **Numbered** check box to number the positions.
10. Select the **Internal terminator** check box if you need to add an internal terminator on the new termination block.
11. In the **Description** field, type a short description for the new configuration.
12. From the **Manufacturer** list, select the appropriate termination block manufacturer. If the required value is not available, click  next to the list arrow to add or edit values for this list.
13. From the **Model** list, select the appropriate termination block model. If the required value is not available, click  next to the list arrow to add or edit values for this list.
14. Use the **Number of apparatuses** spinner or type the required number of apparatuses to be added. This way you can have a block of more than one apparatuses. This number will be displayed by default when adding a new apparatus to an apparatus strip.
15. In the **Apparatus numbering** group box, type the apparatus name in the **Prefix** field if required.
TIP Apparatus names can be 15-character long. This name will appear in the **Domain Explorer** within the new apparatus strip (only the first four characters), in the **Connection** window, and in the appropriate reports.
16. Select the **Numbered** check box to number the apparatuses.
17. In the **Apparatus configuration** group box, from the **Orientation** list, select one of the following terminal patterns:
 - **Left / Right** — an apparatus that has two sides: input and output (that is, left and right).
 - **Left** — an apparatus that has wires on the left side only.
 - **Right** — an apparatus that has wires on the right side only.
 - **Regular** — an apparatus that has a conventional terminal that functions as a single unit.
18. Do the following to define the apparatus terminal name and color:
 - a. Click in the left and/or right terminal side text box and type the required terminal name, for example **1A**:



- a. Select a color of the terminal side from the list, for example red:



19. Repeat steps 17 and 18 for the other terminal side if appropriate. Note that the line between the two terminal list arrows represents the internal relation between the input and output terminals.
20. Click **Add Terminal** to add another terminal to this apparatus configuration if needed and then repeat steps 14 through 16.
21. Click **Delete Terminal** if you need to delete a selected terminal from the current configuration.
22. Click **Save** and then **Close**. You can now add the required apparatus as needed.

Create an Apparatus Group

SmartPlant Instrumentation allows you to add an apparatus group to a panel or a wiring equipment item. An apparatus group constitutes an apparatus strip (that is, a terminal strip) with a number of apparatuses created according to a selected apparatus configuration. You can add an apparatus group to a panel or a wiring equipment item.

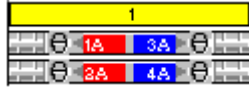
The software adds an apparatus group after selecting an appropriate apparatus configuration. You can add a new apparatus group after creating an apparatus configuration. When adding a new apparatus to a panel, the software also creates a new apparatus (terminal) strip on which the apparatus is created.

You can add an apparatus to a panel or to a wiring equipment item. It is also possible to add a single apparatus to another apparatus.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Do one of the following:
 - In the **Domain Explorer**, double-click the **Panels by Location** folder, and select a panel.
 - In the **Domain Explorer**, double-click the **Panels by Category** folder, and select a panel.
 - In the **Reference Explorer**, double-click the **Panels** folder, and select a panel.
3. Right-click a panel to which you want to add an apparatus and then on the shortcut menu, click **New > Apparatus Group**.
4. On the **Apparatus Group** dialog box, select the desired configuration from the **Configuration name** list.
5. In the **Apparatus profile** group box, in the **Configuration name** field, type the name of the new apparatus configuration.
TIP If you do not have an appropriate configuration, create one. For details, see *Define an Apparatus Configuration* (on page 73).
6. Use the **Number of apparatuses** spinner or type the required number of apparatuses to be added. This way you can have a block of more than one apparatuses. This number will be displayed by default when adding a new apparatus to an apparatus strip.
7. Click **Create**.
TIP SmartPlant Instrumentation automatically assigns sequential position numbers to each apparatus. You can edit them later if required. For details, see *Edit Apparatus Terminal Properties* (on page 76).
8. In the **Terminal Strip Properties** dialog box, define the new apparatus strip as required and click **OK**. For details, see *Create a Terminal Strip* (on page 85).
9. Click **Close** in the **Apparatus** dialog box.

TIP The new apparatus appears in the **Domain Explorer** within the new terminal strip.

10. Connect the apparatus to the appropriate cables. Right-click the apparatus group you want to connect and then on the shortcut menu, click **Actions > Connection**. The **Connection** window opens displaying the apparatus terminals in yellow:



Add an Apparatus to an Apparatus Group

SmartPlant Instrumentation allows you to add a single apparatus to an apparatus group. The software creates a new apparatus without any terminals and adds it at the bottom of the apparatuses in the current apparatus group. If needed, you can then change apparatus sequence within the group and add the required terminals.

1. In the **Domain Explorer** or **Reference Explorer**, right-click an apparatus group.
2. On the shortcut menu, click **New > Apparatus**.
3. In the **Apparatus Properties** dialog box, enter the apparatus properties as you require and click **OK**.

Edit Apparatus Terminal Properties

The following procedure explains how to edit the properties of an apparatus terminal.

1. In the **Domain Explorer**, select the terminal that you want to edit.
2. Right-click the selected terminal and click **Properties** on the shortcut menu.
3. In the **Terminal Properties** dialog box, on the **General** tab, from the **Terminal configuration** list, select the required orientation of the current apparatus terminal.
4. Define the terminal name by typing values in the **Left terminal** and **Right terminal** boxes as you require. The combination of the left and right terminal name will then appear in the **Domain Explorer**.
5. In the **Sequence** field, type a unique terminal sequence number.

TIP This number designates the physical position of the terminal on the strip. The software assigns this number automatically when you add a new terminal to the selected terminal strip. If you change the sequence number of a terminal, be sure to change the sequence numbers of the other terminals on the apparatus strip to avoid duplicate values, which may interfere with the termination process.

6. Define the color as required.
7. Under **Details**, from the **Manufacturer** list, select the required terminal manufacturer. If the required value is not available, click  next to the list arrow to open the **Terminal Manufacturer** dialog box where you can add additional items to the **Manufacturer** list.
8. From the **Model** list, select the required terminal model. If the required value is not available, click  next to the list arrow to open the **Terminal Model** dialog box where you can add additional items to the **Model** list.
9. From the **Type** list, select the appropriate terminal type to which the new terminal belongs. If the required type is not on the list, click  next to the list arrow to open the **Terminal Type** dialog box where you can add additional items to the **Type** list. The **Terminal Type** dialog box, also allows you to replace the default image of terminal side icons for single and multiple connections.
10. In the **Note** text box, type, if required, a brief note or a remark about this terminal.

11. Click **OK** to create the new terminal.

The Plug-and-Socket Group Wizard

This wizard takes you through the steps required to create a plug-and-socket group for wiring equipment. You can then connect cables to this group and display the wiring equipment in a Cable Block Diagram.

To access the wizard, in the Domain Explorer right-click a wiring equipment item and on the shortcut menu click **New > Plug-and-Socket Group**.

Welcome to the Plug-and-Socket Group Wizard

This page appears when you open the **Plug-and-Socket Group** wizard. Click **Next** to continue.

Define the New Ports Group (Plug-and-Socket Group Wizard)

Type a name for you group.

Define Plug-and-Socket Group Layout (Plug-and-Socket Group Wizard)

On this page you define the number of connectors to be created in the group.

Create coupled connectors — Select this check box to create coupled connectors (the internal links are made automatically). Clear this check box if you want to create the connector couplings manually.

Coupled Layout

Only available when you select the **Create coupled connectors** check box.

Number of connectors — Enter the number of connectors you require for one side of the group. The software creates the connectors, and their internal links, on both sides of the group.

TIP You can manually change the internal links later if required.

Non-coupled layout

(Only available when you clear the **Create coupled connectors** check box.)

Number of connectors on the left side — Enter the number of connectors you require on the left side of the group.

Number of connectors on the right side — Enter the number of connectors you require on the right side of the group.

Connector numbering

Prefix — Type a prefix for the connector numbering, if required.

Define the Ports (Plug-and-Socket Group Wizard)

On this page you can select individual connectors and edit their names if required.

Define the Plug-and-Socket Ports (Plug-and-Socket Group Wizard)

On this page you define the connector type for the connectors in the plug-and-socket group. You can define them one-by-one, or define one connector and apply its definitions to all the connectors.

Port — From the list select the connector you want to define.

Connector type — From the list select the type of connector you want to define the selected connector. Click  to open the **Connector Types** dialog box and create a new connector type if required.

Male or Female — From the list select Male or Female.

Connector type definition display — Displays the **Sequence**, **Name**, and **Polarity** of the selected connector type.

Apply to all — Select the check box to apply the selected connectors definitions to all the connectors of the group. Clear the check box if you want to define each connector individually.

Completing the Plug-and-Socket Group Wizard (Plug-and-Socket Group Wizard)

This page appears when the wizard is ready to create the plug-and-socket group from the parameters entered in the wizard.

View and modify the internal links — Select this check box if you want to make changes to the internal links before the wizard creates the plug-and-socket group. Clear this check box if you do not want to change the internal links.

Finish — Click Finish to;

- If you chose to modify the internal links, open the **Plug-and-Socket Group Internal Links** dialog box where you can make your modifications. Clicking OK closes the dialog box and completes creating the new group.
- If you chose not to modify the internal links, the wizard closes and creates the new group in the **Domain Explorer**.

After creating the required plug-and-socket group, you can start connecting cables to the plug-and-socket group connectors.

1. In the **Domain Explorer**, locate the wiring equipment with the plug and socket group.
2. Right-click the wiring equipment and on the shortcut menu, click **Actions > Connection**.

TIPS

- When you have more than one plug-and-socket group associated with the wiring equipment, you can select the specific group from the **Panel** list in the **Plug-and-Socket Box Connection** screen.
 - You can also right-click on the plug-and-socket group  and on the shortcut menu, click **Actions > Connection**.
3. In the **Cables** folder, locate the cable you want to connect to the plug-and-socket group.

NOTE To connect a cable to the plug-and-socket group, the cable must have a plug-and-socket connector on one end of the cable.

4. Drag the cable from the **Domain Explorer** to the required connector in the **Plug-and-Socket Box Connection** dialog box.
5. When you finish connecting all the cables to their connectors, close the **Plug-and-Socket Box Connection** dialog box.

Modify the Internal Links of a Plug-and-Socket Group

This procedure can be used when creating a plug-and-socket group for wiring equipment or for modifying an existing plug-and-socket group.

1. To open the **Plug-and-Socket Internal Links** dialog box, do one of the following:
 - When creating a new plug-and-socket group with the **Plug-and-Socket Group Wizard**, on the **Completing the Plug-and-Socket Group Wizard**, select **View and modify the internal links** and click **Finish**.
 - On an existing plug-and-socket group, right-click and on the shortcut menu click **Properties**. Click **Next** until the **Completing the Plug-and-Socket Group Wizard** page appears, select **View and modify the internal links** and click **Finish**.
2. To disconnect a link, do the following:
 - a. On the **Plug-and-Socket Internal Links** dialog box, press the **Conn.Mode** button to release the button.
 - b. On the display pane select the link you want to disconnect.

TIPS

- You can only select one link at a time.
 - The link turns red when selected, click the link again to clear the selection.
- a. Click **Disconnect**.
 3. To connect a link, do the following:
 - a. On the **Plug-and-Socket Internal Links** dialog box, press the **Conn.Mode** button to select the button.

TIP The cursor changes to a dot (•) over the display pane when the connection mode is selected.

- b. Click on the empty pin of the left side connector you want to link from.
- c. Click on the empty pin of the second connector (right side) to create the link.

4. When you have finished your modifications, click **OK**.

SECTION 8

Terminal Strips and Terminals

This section deals with the creation and management of terminal strips and terminals.

In This Section

Terminal Strips.....	81
Define a Terminal Strip Configuration	81
Modify a Terminal Strip Configuration	83
Terminal Strip Configuration Examples	84
Create a Terminal Strip.....	85
Move a Terminal Strip to Another Parent Item	85
Add a Terminal to a Terminal Strip.....	86
Add a Group of Terminals to a Terminal Strip	87
Edit the Properties of a Terminal	87
Move a Terminal	88
Select a Graphic for a Terminal Connection.....	89

Terminal Strips

In SmartPlant Instrumentation, you create terminal strips based on a particular configuration of terminals. This configuration is a repeating numbering pattern that enables logical numbering of terminals on a strip. A strip terminal configuration includes the total number of terminals and the terminal numbering system.

You can create terminal strips either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a terminal strip that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference terminal strips facilitate fast creation of terminal strips in your plant by copying a terminal strip from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a terminal strip that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the terminal strips in the current <unit>.

In the **SmartPlant Instrumentation Explorer**, terminal strips are child items of panels, racks, slots, and various wiring equipment items. Terminal strips are parent items of channels and terminals.

Define a Terminal Strip Configuration

When creating a terminal strip, you are required to use a basic repeating numbering pattern (configuration) to enable logical numbering of terminals. A numbering pattern identifies the basic numbering system, or stepping method used for terminals in a terminal strip. Set +1,-1,+2,-2, for example, has a pattern of two, while the Set 1,2,3SH,4,5,6SH has a pattern of three.

You can label the terminals on a terminal strip according to any numbering system that you require (up to ten characters in length). To facilitate logical terminal numbering, SmartPlant Instrumentation requires you to configure the basic repeating numbering pattern (configuration) for the terminals in the new terminal strip before defining that terminal strip.

When creating a strip in an I/O card or I/O termination, you use configurations that include channels. If you want to create a terminal strip without channels in one of these panels, set the number of terminals per channel to zero.

1. In the **Domain Explorer** or **Reference Explorer**, right-click a panel.
2. On the short-cut menu, click **New > Terminal Strip**.
3. On the **Terminal Strip Configuration** dialog box, click **New**.
4. On the **Number of Terminals in Pattern** dialog box, type or select a number to define the number of terminals to be repeated in the pattern and click **OK**.

TIPS

- This number determines the number of rows (terminals) that will constitute the basic repeating numbering (naming) pattern.
 - This numbering pattern is shown in the **Terminal Numbering Pattern** data grid on the **Terminal Strip Configuration** dialog box.
5. Under **Configuration name**, type the name of the new configuration.
 6. Under **Configuration description**, type an optional description of the new configuration.
 7. Type or select the total number of terminals required in the terminal strip.
 8. Select the starting number for the first terminal in the terminal strip.
 9. Select the required number of terminals per channel or segment.
 10. Select the first channel.
TIP If the new configuration is to be used to create I/O cards, the value in the **Number of terminals per channel** and **First channel** fields must be larger than zero. Otherwise, the terminals on the terminal strip are created without channels and you will not be able to create an I/O card.
 11. In the **Terminal Numbering Pattern** data grid, click and type in the **Prefix** field, click and type a value if you want to prefix the names of the terminals.
 12. Select the **Incremented** check box if you want the terminal number to be incremented within the pattern.
 13. Select the **Numbered** check box if you want the number of the terminal to appear between the prefix and the suffix.
 14. In the **Suffix** field, click and type a value if you want a suffix for the names of the terminals.
 15. From the **Orientation** list, select the connection orientation to the terminal:
 - **Regular** — a conventional terminal that functions as a single unit.
 - **Left** — a terminal that has wires on the left side only.
 - **Right** — a terminal that has wires on the right side only.
 - **Left / Right** — a terminal that has wires on the left side only.
 16. Click **Save**.

NOTES

- If the required number of terminals is not exactly divisible by the number of terminals in a pattern, the remaining terminals are labeled SPARE.
- For examples of possible terminal strip configurations, see *Terminal Strip Configuration Examples* (on page 84).

Modify a Terminal Strip Configuration

The following procedure explains how to modify an existing configuration of terminals. Modifying a terminal configuration does not affect the existing terminal strips. They retain their previous terminal configuration.

1. In the **Domain Explorer** or **Reference Explorer**, right-click a panel.
2. On the short-cut menu, click **New > Terminal Strip**.
3. On the **Terminal Strip Configuration** dialog box, select a configuration from the **Configuration name** list.
4. Click **Edit**.

TIP The software does not allow you to change the terminal numbering pattern of an existing configuration. Therefore, you have to define a new configuration if you need to make such a change.

5. Under **Configuration name**, type the name of the new configuration.
6. Under **Configuration description**, type an optional description of the new configuration.
7. Type or select the total number of terminals required in the terminal strip.
8. Select the starting number for the first terminal in the terminal strip.
9. Select the required number of terminals per channel or segment.
10. Select the first channel.

TIP If the new configuration is to be used to create I/O cards, the value in the **Number of terminals per channel** and **First channel** fields must be larger than zero. Otherwise, the terminals on the terminal strip are created without channels and you will not be able to create an I/O card.

11. In the **Terminal Numbering Pattern** data grid, click and type in the **Prefix** field, click and type a value if you want to prefix the names of the terminals.
12. Select the **Incremented** check box if you want the terminal number to be incremented within the pattern.
13. Select the **Numbered** check box if you want the number of the terminal to appear between the prefix and the suffix.
14. In the **Suffix** field, click and type a value if you want a suffix for the names of the terminals.
15. From the **Orientation** list, select the connection orientation to the terminal:
 - **Regular** — a conventional terminal that functions as a single unit.
 - **Left** — a terminal that has wires on the left side only.
 - **Right** — a terminal that has wires on the right side only.
 - **Left / Right** — a terminal that has wires on the left side only.

16. Click **Save**.

NOTES

- If the required number of terminals is not exactly divisible by the number of terminals in a pattern, the remaining terminals are labeled SPARE.
- For examples of possible terminal strip configurations, see *Terminal Strip Configuration Examples* (on page 84).

Terminal Strip Configuration Examples

Each of the following examples demonstrates how various features of the strip configuration operate. In each case, the terminal strip has a repeating sequence of three terminals, and a total of 8 terminals (the last 2 terminals are designated SPARE, and are not shown).

Input Data and Explanation

Terminal Numbering Pattern			
Prefix	Incremented	Numbered	Suffix
+	☐	☐	☐
-	☐	☐	☐
SHD	☐	☐	☐

No numbering

Terminal Numbering Pattern			
Prefix	Incremented	Numbered	Suffix
+	☐	☒	☒
-	☐	☐	☒
SHD	☐	☐	☐

The + and – terminals are numbered.
The numbers are incremented for the first terminal of each repeating sequence.

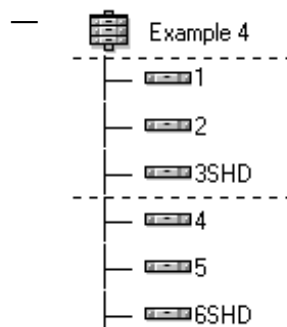
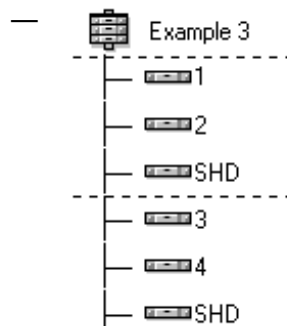
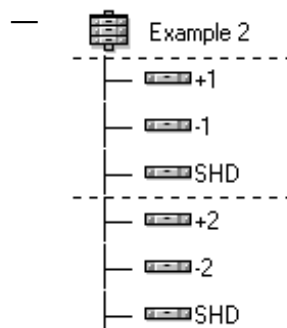
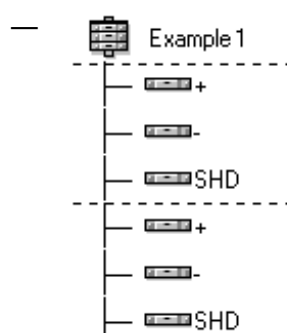
Terminal Numbering Pattern			
Prefix	Incremented	Numbered	Suffix
	☒	☒	☐
	☒	☒	☐
SHD	☐	☐	☐

The + and – terminals are numbered.
The numbers are incremented for all the numbered terminals.

Terminal Numbering Pattern			
Prefix	Incremented	Numbered	Suffix
	☒	☒	☒
	☒	☒	☒
	☒	☒	☒
SHD	☐	☐	☐

The + and – terminals are numbered.
All the numbers are numbered and incremented.
Note the use of the suffix.

Resultant Terminal Configuration



Create a Terminal Strip

This procedure explains how to create a terminal strip. You can add a terminal strip to a panel, rack, slot, or a wiring equipment item.

1. In the **Domain Explorer** or **Reference Explorer**, right-click one of the following items:
 - Panel
 - Rack
 - Slot
 - Wiring equipment item, for example an I/O card
2. On the short-cut menu, click **New > Terminal Strip**.
3. On the **Terminal Strip Configuration** dialog box, select a configuration from the **Configuration name** list.
TIP If you do not have an appropriate terminal configuration, click **New**.
4. Click **Create**.
5. On the **Terminal Strip Properties** dialog box, on the **General** tab, under **Terminal strip** type the name of the new terminal strip.
6. From the **Type**, **Model**, and **Manufacturer** lists, select appropriate values to define the new terminal strip as you require. If any of the lists do not contain the value that you need, click  to open the relevant supporting table where you can manage the contents of the select lists.
7. Type a value for mounting, dimensions, and rail if needed.
8. Under **Sequence**, enter a number to define the sequence of the current terminal strip in its parent item (a panel, rack, slot, or wiring equipment).
9. To manage the revisions of the terminal strip, click **Revisions**.
10. Click the **Associate Symbols** tab to associate a symbol with the current terminal strip. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **OK** to create the new terminal strip.

Move a Terminal Strip to Another Parent Item

This option enables you to move a terminal strip with connected cables to another parent item (panel, rack, slot, or wiring equipment). The software retains all the connection definitions after moving a terminal strip.

1. In the **Domain Explorer** or **Reference Explorer**, select a terminal strip that you want to move.
2. Expand the hierarchy of the target parent item.

TIP You can open another instance of the **Domain Explorer** or **Reference Explorer** and place it beside the one that is already open to make the drag-and-drop operation easier.

3. Drag the selected terminal strip to the desired parent item.

NOTE The following restrictions apply when trying to move a terminal strip with connected cables:

- The terminal strip is associated with an instrument type.
- The terminal strip is used by an auto-wiring task.
- The terminal strip has connected terminals or channels that are associated with control system tags.

- The terminal strip has not been claimed for a project.
- The terminal strip is associated with a local or general signal.
- The target parent item is a plug-and-socket box.

Add a Terminal to a Terminal Strip

This procedure explains how to add a single terminal to an existing terminal strip.

1. In the **Domain Explorer** or **Reference Explorer**, select a terminal strip to which you want to add a terminal.
2. Right-click the selected terminal strip and then on the shortcut menu, click **New > Terminal**.
3. On the **General** tab of the **Terminal Properties** dialog box, accept or select the required terminal orientation.
4. Enter the terminal number or numbers in the **Left terminal** or **Right terminal** fields.
TIP Your selection of the terminal orientation determines whether the **Left terminal** or **Right terminal** fields are available. If you select **Left and right-oriented**, the software lets you enter a value for both left and right terminals. If you select **Regular**, the software lets enter the terminal number under **Terminal**.
5. Under **Sequence**, type a unique terminal sequence number.
TIP The sequence number designates the physical position of the terminal on the terminal strip. The software assigns a sequence number automatically when you add a new terminal to a terminal strip. If you change the sequence number of a terminal, be sure to change the sequence numbers of the other terminals on that terminal strip to avoid duplicate values, which may interfere with the termination process.
6. Select a terminal color as needed.
7. In the **Details** group box, enter a channel number.
8. If the current terminal is in a channel which in turn is a wiring equipment sub-item, enter a sequence number with in the current channel.
9. From the **Type**, **Model**, and **Manufacturer** lists, select appropriate values to define the new terminal as you require. If any of the lists do not contain the value that you need, click  to open the relevant supporting table where you can manage the contents of the select lists.
10. In the **Note** text box, type, if required, type a brief note or a remark about this terminal.
11. Click the **Associate Symbols** tab to associate a symbol with the current terminal. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
12. Click **OK** to create the new terminal.

Add a Group of Terminals to a Terminal Strip

This procedure explains how to add a new group of terminals to an existing terminal strip.

1. In the **Domain Explorer** or **Reference Explorer**, select a terminal strip to which you want to add a group of terminals.
2. Right-click the selected terminal strip and then on the shortcut menu, click **Actions > Add Terminal Group**.
3. On the **Add Terminals** dialog box, select an appropriate terminal configuration from the **Configuration name** list. The configuration you choose determines the labeling of the terminals you are about to create.

TIP If the required configuration is not available, close this dialog box and create a new terminal configuration.

4. Do one of the following:
 - If the selected configuration is suitable, click **OK** to add the terminals.
 - Change the configuration according to your requirements as specified in the steps below.
5. In the **Total number of terminals** spin box, enter the total number of new terminals required.
6. In the **First terminal of added group** spin box, enter the number of the next incremented terminal.

TIP To continue the numbered sequence of the existing terminals, you should enter the next numerical value of the terminal, and not the ordinal sequence. For example, if you already have 48 terminals arranged in groups of three, where the last 3 terminal labels are +16, -16, and SH, the value in the **First terminal of added group** spin box must be 17 and not 49.

7. In the **Terminal Numbering Pattern** group box, do the following:
 - a. Type a prefix in the **Prefix** field if required.
 - b. Select the **Incremented** check box if you want the terminal number to be incremented within the pattern.
 - c. Select the **Numbered** check box if you want the number of the terminal to appear between the prefix and the suffix.
 - d. Type a suffix if required.
8. Click **OK** to create the new group of terminals on the terminal strip.

NOTE This dialog box does not allow you to save the changes you make to an existing terminal strip configuration. The software applies the changes only to the group of terminals you are currently adding.

Edit the Properties of a Terminal

1. In the **Domain Explorer** or **Reference Explorer**, select a terminal that you want to edit.
2. Right-click the selected terminal and then on the shortcut menu, click **Properties**.
3. On the **General** tab of the **Terminal Properties** dialog box, accept or select the required terminal orientation.
4. Enter the terminal number or numbers in the **Left terminal** or **Right terminal** fields.

TIP Your selection of the terminal orientation determines whether the **Left terminal** or **Right terminal** fields are available. If you select **Left and right-oriented**, the software lets you enter a value for both left and right terminals. If you select **Regular**, the software lets enter the terminal number under **Terminal**.

5. Under **Sequence**, type a unique terminal sequence number.

TIP The sequence number designates the physical position of the terminal on the terminal strip. The software assigns a sequence number automatically when you add a new terminal to a terminal strip. If you change the sequence number of a terminal, be sure to change the sequence numbers of the other terminals on that terminal strip to avoid duplicate values, which may interfere with the termination process.

6. Select a terminal color as needed.
7. In the **Details** group box, enter a channel number.
8. If the current terminal is in a channel which in turn is a wiring equipment sub-item, enter a sequence number with in the current channel.
9. From the **Type**, **Model**, and **Manufacturer** lists, select appropriate values to define the new terminal as you require. If any of the lists do not contain the value that you need, click  to open the relevant supporting table where you can manage the contents of the select lists.
10. In the **Note** text box, type, if required, type a brief note or a remark about this terminal.
11. Click the **Associate Symbols** tab to associate a symbol with the current terminal. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
12. Click **OK** to create the new terminal.

Move a Terminal

This option enables you to move a terminal from one position to another on a terminal strip/channel, or from one terminal strip or channel to another terminal strip or channel.

IMPORTANT It is possible to move a terminal only if the following is true:

- The source terminal is not connected to any wiring.
- When the terminal sequence is changed, even if the terminal is connected.
- When the terminal is connected and has a signal, and the terminal is being moved to a different terminal strip provided the target terminal strip is under wiring equipment that is not an I/O card or I/O termination.
- When the terminal is connected but does not have a signal and is being moved from one terminal strip to another.
It is **not** possible to move a connected terminal that has a signal or if the target terminal strip is under an I/O card or I/O termination.

1. In the **Domain Explorer** or **Reference Explorer**, select a terminal that you want to move.
2. Expand the hierarchy of the target terminal strip.

TIP You can open another instance of the **Domain Explorer** or **Reference Explorer** and place it beside the one that is already open to make the drag-and-drop operation easier.

3. Drag the selected terminal to the desired terminal strip.

Select a Graphic for a Terminal Connection

This procedure explains how to replace the default screw head graphic that represents a terminal connection of a terminal in the **Connection** window and various Panel-Strip reports. You select a graphic for a particular terminal type and the number of wires connected to the terminal connection. That is, a specific graphic represents a connector type and whether there is no wire, a single wire, or multiple wires, connected on that particular side of the terminal.

1. On the **Wiring Module** window menu bar, click **Tables > Terminal > Types**.
2. On the **Terminal Types** dialog box, select a terminal type or create a new one.
3. Click in the **G1** box to select a graphic that will represent a terminal connection with a single wire or without any wire.
4. Click in the **G2** box to select a graphic that will represent a terminal connection with a multiple wires.

NOTES

- The selected graphic appears in the **G1**, **G2**, or both boxes. The size of the **G1** and **G2** boxes represents the actual size of the icons that appear in the **Connection** window.
- If you do not select anything in the **G1** or **G2** box, SmartPlant Instrumentation displays the default gray screw head  for terminal connections with or without single wires and the turquoise screw head  for terminal connections with multiple wires.

SECTION 9

Making Connections in SmartPlant Instrumentation

SmartPlant Instrumentation employs two methods for implementing cable terminations – direct visual connection, and a more advanced technique called **connection type**. The connection concept behind these methods is based on the simple idea of letting you work as you would in the field, meaning that if you install a wire and connect it to a panel, for example, you can do exactly the same on the screen.

The software enables you to make connections by using the drag-and-drop operation in the **Connection** window.

You can make the following connections:

- Connect cables and cable sets to terminals in **single** mode — this option allows you to connect cables one by one to the appropriate terminals.
- Connect cables and cable sets to terminals in **batch** mode — this option allows you to simultaneously connect cables to the appropriate terminals and define their connections.

NOTES

- You can keep more than one **Connection** window open at the same time. This can be useful when viewing the wiring routing.
- Dragging a cable, cable set, or wire to a terminal not currently visible in the **Connection** window makes the screen scroll up or down until the required terminal is displayed.
- If the width of your computer monitor allows it, you can follow the course of a cable from one terminal strip to the other (provided that the cable is connected between two terminal strips). You can view the cable continuation between two terminal strips by resizing the **Connection** window and clicking  (or clicking **Actions > Adjacent** on the **Connection** window menu bar) to simultaneously display both terminal strips.
- When connecting the right side of a DCS or PLC panel, the software increases the signal level by one.

Define a Connection Type

This option allows you to select, create, or edit a connection type that facilitates rapid cable set connections. You define a connection type that contains a connection pattern. A connection type contains the definition of the pattern by which wires will be connected on a terminal strip, and allows automatic creation and connection of jumpers between successive cable sets. Once a wiring pattern has been defined, it can be selected whenever desired and will determine how the connections are made.

1. Start the **Wiring** module and then do one of the following:
 - Click **Actions > Connection Types**.
 - Click  on the Wiring module toolbar.
2. On the **Connection Type** dialog box, click **New**.
3. On the **New Connection Type** dialog box, configure the new connection type by selecting the number of wires and jumpers that the new connection type will contain.
4. Click **OK** to return to the **Connection Type** dialog box.
5. In the **Connection type** text field, type the name of the new connection type.
6. In the **Description** text field, type a short description if needed.
7. In the **Skip between sets** field, enter the number of terminals that will be skipped between the first terminals of each cable set.

8. In the **Wires** group box, do the following:
 - a. Click in the **Skip** text field and type the number of terminals to skip before connecting this wire in the cable set. The skip is always relative to the first terminal to which a wire in the cable set is connected. Enter the number of terminals to be skipped according to your needs. In most cases, this value is incremented for successive wires. You can accept the displayed value if suitable.
 - b. In the **Side** field, click the **A — Active** or **O — Opposite** option button to select the terminal side to which you want to connect each wire. The side that is currently active depends on the side you start to connect.
 - c. Click in the **Land** field to connect a wire. A screw head  appears indicating that a connection has been configured.

TIP The **Wire** field displays a specific wire in the cable set. The displayed **W00x** values represent the wire names and are used to illustrate the current connection type configuration. This field is not editable.

9. To configure a jumper connection, do the following in the **Jumpers** group box:
 - a. From the **Type** list, select a jumper type from this list. The values in this list are determined in the **Wire Type** supporting table.
 - b. From the **Color** list, select the required jumper color.
 - c. Click in the **End 1** and **End 2** fields to connect to a jumper. A screw head  shows up indicating that a connection has been configured.
 - d. Click in the **Skip** text field and type the number of terminals to skip for the jumper in the cable set for landing. You can accept the displayed value if appropriate. Repeat the same procedure for the other end.
 - e. In the **Side** fields, click the **A – Active** or **O – Opposite** option button to select the terminal side where the jumper will be connected. The side that is currently active depends on which side you start to connect.

TIPS

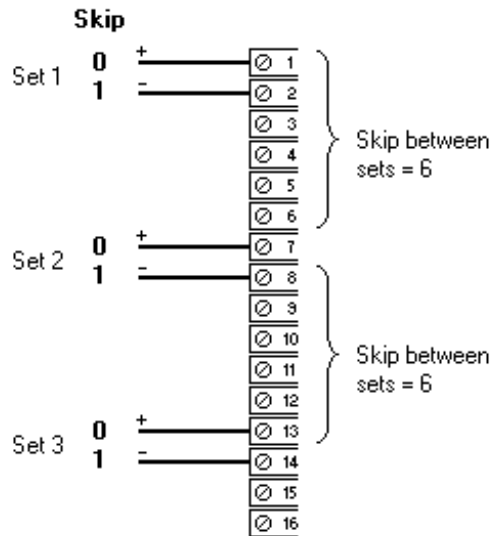
- The options in the **Jumpers** group box become available only if you have selected jumpers to be configured on the **New Connection Type** dialog box.
- The terminals to which the ends of a jumper are connected must reside on the same strip. Both ends of a jumper must be connected for it to appear in the display region. For an illustration, see *Connection Type Examples* (on page 93).

10. Click **Save**.

Connection Type Examples

Example 1: (a two-wire cable set)

The following connection type example illustrates how the connections are made for a 2-wire cable set when the skip between cable sets is 6:



The position of the + wire in each cable set to be connected has been assigned a skip of 0. The - wire of Set 1 has been assigned a skip of 1. The - wires of Sets 2 and 3 have each been assigned a skip of 6.

Example 2: (two wires with a jumper)

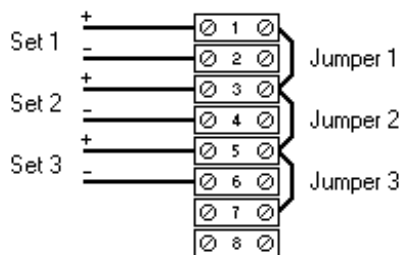
The following example illustrates how you can connect a single pair to a strip and create a jumper on the opposite side between the first and third terminals. The next pair will be connected to the third terminal, if the skip between sets parameter was set to 2.

Wire	Skip	Side	Land
W1	0	● A ○ 0	⊗
W2	1	● A ○ 0	⊗

W1	0	⊗	J1
W2	1	⊗	J1
	2	⊗	J1

Jump	Type	Color	End 1	Skip	Side	End 2	Skip	Side
J1		RD	⊗	0	○ A ● 0	⊗	2	○ A ● 0

The outcome of this operation, when terminating multiple pairs, will be pairs connected sequentially where every first wire in a set is connected by a jumper to the one that follows, as illustrated below:



Connect a Cable or a Cable Set to a Terminal Strip

This procedure explains how to connect a cable or cable set to a terminal strip. The connection is accomplished by dragging a cable or cable set to a terminal in the **Connection** window. Then, you are required to define the cable connection, which effects the connection. Note that you can connect only plant cables and panels and not reference panels and cables. However, while the **Connection** window is open, you can copy reference items from the **Reference Explorer** to the **Domain Explorer** and then drag them to the **Connection** window.

1. Press F7 to open the **Domain Explorer**.
2. Expand the **Cables** folder or a panel hierarchy and then right-click one of the following items:
 - A cable
 - A cable set
 - A panel
 - A terminal strip
3. On the shortcut menu, click **Actions > Connection**.
4. In the **Connection** window, from the **Terminal strip** list, select an appropriate terminal strip.

TIP If the terminal strip that you need does not exist in the **Domain Explorer**, you can copy a reference terminal strip from **Reference Explorer** to the **Domain Explorer** while the **Connection** window is open. (Press F8 to open the **Reference Explorer**.)
5. In the **Domain Explorer**, select the cable or cable set that you want to connect.
6. Drag the selected cable or cable set to the appropriate terminal in the data window.
7. Drop the cable or cable set on the required terminal (shown as a screw head ) to open the **Cable Connection Definition** dialog box.
8. On the **Cable Connection Definition** dialog box, do the following:
 - a. Select **End 1** or **End 2** for the side of the cable that you want to connect.
 - b. To change the terminal number where the first wire is to be connected, from the **Start at terminal** list, select the required terminal.
 - c. In the **Cable set connection details** data window, select the cable sets that you want to connect, or choose **Select all cable sets**.
 - d. From the **Connection Type** list, select the connection type for each cable set that you are going to connect.

- e. Select **L** (left) or **R** (right) for the physical side of the terminal where the cable set enters. Note that screw head  indicates that the cable set is already connected on the selected cable end.
- f. Click **Connect** to connect the selected cable sets and return to the **Connection** window.

NOTES

- SmartPlant Instrumentation connects the individual wires automatically in accordance with the selected connection type.
- You can double-click a cable, cable set, wire, terminal, or terminal strip header in the **Connection** window to edit it.

Connect Device Cables to a Terminal Strip in Batch Mode

Use this procedure to connect device cables or cable sets to a terminal strip in batch mode. Note that these are device cables that are already connected to device panels.

1. Press F7 to open the **Domain Explorer**.
2. Expand the **Panels by Location** or the **Panels by Category** folder to display the existing panels and terminal strips.
3. Right-click a terminal strip to which you want to connect device cables, and then on the shortcut menu, click **Actions > Batch Connection**.
4. In the **Domain Explorer**, select the device cables that you want to connect and drag them to the **Batch Device Cable Connection** window.

TIPS

- You can only drag those cables that have been connected to device panels on one end and unconnected on the other end. Also, there cannot be any connectors on the unconnected end.
 - You can filter the display of cables in the **Domain Explorer** so that you can easily find the device cables that you need. For details, see *Pre-Assign Junction Boxes to Device Panels* (on page 125).
 - Click  on the lower toolbar to open a pop-up window that displays which terminals are connected.
 - The settings displayed under **Starting Terminal** are determined by the automatic implementation of connection type definitions for the cables that you move to the data window.
 - You can change the order of cables in the data window by dragging them up or down.
5. To change the connection type for a given cable set, from the Connection Type list, select an appropriate connection type.
 6. When changing the connection type for the first cable set, in response to the 'Do you want to change the connection type for all cable sets?' prompt, do one of the following:
 - To change the connection type for all of the cable sets displayed in the data window, click **Yes**. Note that this also results in automatic renumbering of the starting terminals for all of the cable sets.
 - To change the connection type for the given cable set only, click **No**. Note that this results in no automatic renumbering of starting terminals.
 7. To change the cable set starting terminals, do one of the following:
 - To change the starting terminal for a given cable set, select a value from the **Starting Terminal** list.
 - To automatically renumber starting terminals according to cable type definitions, from the selected cable set and below, click **Actions > Reassign**.

8. Under **Terminal Side** accept the default setting **L** to connect the device cables to the left side of the panel, or select **R** to connect the cables to the right side.
9. Under **Connect all shields to**, do one of the following:
 - Do not select any terminal. Use this option to connect the shields according to the configuration of the active cable types.
 - Select a common terminal to which you connect all of the shields.
10. Under **Connect all overall shields to**, do one of the following:
 - Do not select any terminal. Use this option to connect the overall shields according to the configuration of the active cable types.
 - Select a common terminal to which you connect all of the overall shields.
11. Do one of the following:
 - To prepare for the connection of the all of the device cables displayed in the data window, clear **Connect selected rows only**.
 - To prepare for the connection of some of the device cables displayed in the data window, select the rows that you want to connect and then select **Connect selected rows only**.
12. To connect the devices cables to the terminal strip according to your settings, do one of the following:
 - Click **Actions > Connect**.
 - On the lower toolbar, click .

Connect a Wire to a Terminal Strip

This procedure explains how to connect a wire to a terminal strip.

1. Press F7 to open the **Domain Explorer**.
2. Expand the **Panels by Category** or **Panels by Location** folder.
3. Right-click a panel or a terminal strip and then on the shortcut menu, click **Actions > Connection**.
4. If, in the **Domain Explorer**, you selected a panel and not a terminal strip, in the **Connection** window, select one from the **Terminal Strip** list.
5. In the **Domain Explorer**, select a cable and expand its hierarchy to display its wires.
6. Select a wire and drag it to a terminal (shown as a screw head ) in the **Connection** window.
TIP Click  to find a specific terminal or the next available terminal. SmartPlant Instrumentation marks your terminals with .
7. On the **Wire Connection Options** dialog box, select the wire that you require and then click **OK**.

NOTES

- If you are connecting more than one wire in a cable or cable set to a group of terminals on a terminal strip, make sure that the wire end that you connect is the same for all the wires. This is important to ensure that the connections appear correctly in reports.
- If one end of the wire is already connected, SmartPlant Instrumentation selects the loose end by default.

Move Cables, Cable Sets, and Wires

After making initial connections to terminals, SmartPlant Instrumentation lets you relocate cables, cable sets, and wires to another terminal on the same terminal strip. You can move a wiring item using the drag-and-drop feature.

1. In the **Connection** window, select the appropriate cables, cable sets, or wires.
2. Drag the selected items to another terminal. If you are moving a cable or a cable set, the **Cable Connection Definition** dialog box opens. If you are moving a wire, the **Wire Connection Options** dialog box opens.
3. Select the required definitions on the **Cable Connection Definition** dialog box or on the **Wire Connection Options** dialog box and then click **Connect** or **OK** respectively.

NOTES

- The **Connection** window reopens showing the new connections.
- You should ensure that there are sufficient terminals to accommodate all the wires that you want to move.
- You can connect any number of wires to a single terminal.
- If you are connecting more than one wire in a cable or cable set to a group of terminals, make sure that the wire end that you connect is the same for all the wires. This is important to ensure that the connections appear correctly in reports.

Disconnect Cables, Cable Sets, and Wires

This procedure shows how to disconnect a given cable or selected cable sets and wires from a terminal strip.

1. In the **Connection** window, select the cable, cable sets, or wires you want to disconnect.
TIP SmartPlant Instrumentation automatically selects the wiring items in the hierarchy below the item that you select.
2. In the **Connection** window, do one of the following:
 - Right-click the selected item then click **Disconnect** on the shortcut menu.
 - On the menu bar, click **Connection > Disconnect**.
 - Click  on the **Connection** toolbar.

NOTES

- If a terminal strip or terminal is connected to more than one cable, only wires which are unique to a terminal or which belong to a cable at the top level layer (at the front) will be disconnected by SmartPlant Instrumentation.
- SmartPlant Instrumentation uses  to represent terminals on which multiple wires are connected.
- To ensure disconnection of the correct cables or cable sets, first move the cable you want to disconnect to front by selecting it from the **Cable name** list.

Add Jumpers to the Existing Wiring

SmartPlant Instrumentation enables you to create and connect multiple jumpers in a repetitive way. You can define constant gaps between the jumped terminals if required. Also, you can jumper the last remaining terminal in the selected terminal pattern if required. Note that unlike regular wires, jumpers do not propagate signals. You can assign a wire group to a jumper manually on the **Terminal Connection** dialog box.

1. Press F7 to open the **Domain Explorer**.
2. Expand the **Panels by Location** or the **Panels by Category** folder to display the existing panels and terminal strips.
3. Right-click a panel or a terminal strip and then on the shortcut menu, click **Actions > Connection**.
4. In the **Connection** window, select a wire connected to the terminal that will be the first terminal connected to the first jumper.
TIP Do not make any selection if you want the terminal connected to the first wire to be the connected to the first jumper.
5. In the **Connection** window, do one of the following:
 - Right-click the selected wire and then on the shortcut menu, click **Connect Jumpers**.
 - Click **Connection > Connect Jumpers**.
 - Click .
6. On the **Connect Jumpers** dialog box, do the following:
 - a. From the **Terminal one – first jumper** list, select a terminal and set its **Left** or **Right** orientation.
 - b. From the **Terminal two – first jumper** list, select a terminal and set its **Left** or **Right** orientation.
 - c. If you need to define a constant gap between the jumped terminals, enter a value in the **Interval between jumpers** box.
 - d. From the **Last terminal** list, select the last terminal in the pattern that you want to jumper.
 - e. Select the **Jumper the last terminal** check box if you want to jumper the last remaining terminal in the terminal pattern.
 - f. Select a jumper color.
 - g. In the **Jumper name** box, type the name of the new jumper, or accept the default setting. By default, SmartPlant Instrumentation suggests a name for a new jumper of the format **J<first terminal name>/<second terminal name>**. You can change the preferences so as not to display a suggested jumper name. For details, see *Set Preferences for Automatic Naming of Jumpers* (on page 99).
 - h. Click **OK**.

NOTE Your new jumper appears in the **Connection** window.

Set Preferences for Automatic Naming of Jumpers

1. Click **File > Preferences**.
2. Expand the **Wiring** hierarchy in the tree view and select **Jumpers**.
3. Under **Automatic naming**, select **Name jumpers automatically**.
4. Click **OK**.

Remove a Jumper from the Existing Wiring

This procedure explains how to remove a jumper.

1. In the **Connection** window, select a jumper you want to remove.
2. Do one of the following:
 - Right-click the selected jumper and then on the shortcut menu, click **Disconnect**.
 - On the **Connection** window menu bar, click **Actions > Disconnect**.
 - Click .

NOTES

- Jumpers exist in the database as long as they are connected. Disconnected jumpers are deleted from the database.
- You cannot move a jumper. Instead, disconnect the jumper that you do not need and then add a new jumper as required.

Assigning Wire Designation

The **Terminal Connection** dialog box opens when you click a screw head  on the terminal in the **Connection** window. This dialog box shows the Wire Group, Group Level, and Group Sequence of the wires connected to the selected terminal.

When you click Wire Group, Group Level, or Wire Sequence data, you can reassign the wire designation by selecting the required value from the appropriate lists.

You can assign a new signal name to more than one terminal without having to browse through the **Wire Group** list for each terminal. You use this feature only with terminals whose Wire Group is currently undefined (that is, the **Wire Group** data field displays asterisks).

First, you need to assign a **new** signal name to a terminal whose Wire Group is currently undefined. After assigning the new signal to the current terminal, select another terminal (click the up or down arrow or select a terminal in the **Terminal name** list). If you now click in the **Wire Group** data field, the list opens with the new signal name selected by default. Clicking the terminal name enables you to navigate quickly.

NOTE If you want to reassign wire designations on the **Terminal Connection** dialog box, it is advisable to plan your changes in advance on paper, to avoid conflicting Wire Group, Group Level, and Group Sequence data.

Updating Connection Data

Multiple users can work on the same module. In most cases, you may not edit data that is being edited by another user, however, under certain circumstances, other users may have made changes that affect the data in the **Connection** window. For example, another user has added a new plant cable. This cable does not appear in the **Domain Explorer** until you refresh the window display. Click  to refresh the displayed connection data.

The **Connection** window is automatically refreshed only when you open it. However, SmartPlant Instrumentation does not refresh the data in the **Connection** window, if you have several instances of the **Connection** window open and you switch from one **Connection** window to another.

View Cable Connection Destinations

This option enables you to view the connection destinations of a cable that you select in the **Domain Explorer**. When you select this option, you see the junction boxes and terminals that the selected cable is connected to.

1. In the **Domain Explorer**, double-click the **Cables** folder.
2. Right-click a cable.
3. On the shortcut menu, click **Actions > Destination**.

NOTE The **Cable Destination** pop-up window displays individual connections for the selected cable. Note that each line shows a connection to a different panel.

Display Adjacent Connections of a Cable

When you are working on wiring connections for a specific panel, you can follow the wiring routing to an adjacent panel.

1. In the **Connection** window, select a cable, cable set, or wire whose routing you want to trace on a particular terminal side.
TIP If you select a cable or cable set, SmartPlant Instrumentation displays the adjacent connection for the first wire. You can follow the routing for the other wires by selecting other panels from the list.
2. Do one of the following:
 - On the menu bar, click **Actions > Open Adjacent Connection**.
 - Right-click the selected item and then on the shortcut menu, click **Open Adjacent Connection**.
 - Click  on the lower toolbar.

NOTE The **Connection** window opens, showing the adjacent panel which is the continuation of the wiring routing.

Display the Farthest Connection of a Cable

While working in the **Connection** window, in addition to displaying the adjacent connection of a cable or a wire, you can also open the farthest connection. This means that the software switches to the end of the wiring routing of the current wire and displays the last panel on that end. If a signal exists in the direction that you chose to switch to, SmartPlant Instrumentation follows the signal to the end. If there is no signal, the software shows the panel that is connected at the end of the wiring routing.

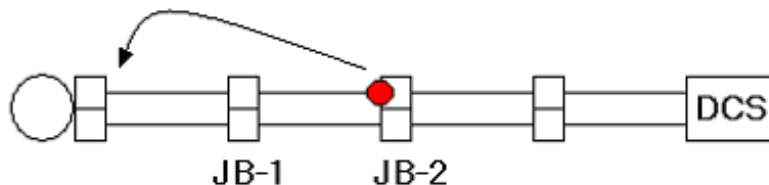
1. In the **Connection** window, select a wire whose routing you want to trace on a particular terminal side.
2. Do one of the following:
 - On the menu bar, click **Actions > Open Farthest Connection**.
 - Right-click the selected item and then on the shortcut menu, click **Open Farthest Connection**.

NOTE The **Connection** window opens, showing the farthest panel at the end of the wiring routing. For examples, see *Examples of Farthest Connections of Cables* (on page 101).

Examples of Farthest Connections of Cables

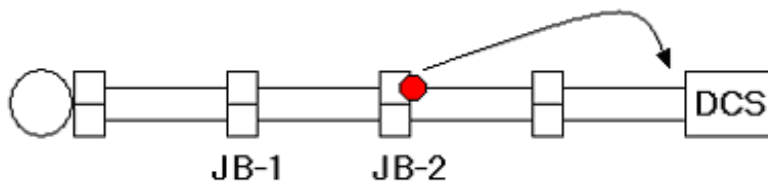
While working in the **Connection** window, in addition to displaying the adjacent connection of a cable or a wire, you can also open the farthest connection. This means that the software switches to the end of the wiring routing of the current wire and displays the last panel on that end. If a signal exists in the direction that you chose to switch to, SmartPlant Instrumentation follows the signal to the end. If there is no signal, the software shows the panel that is connected at the end of the wiring routing.

Example 1:



In this example, you selected the left side connections of JB- 2. Therefore, SmartPlant Instrumentation will display the connection of the device panel.

Example 2:



In this example, you selected the right side connections of JB- 2. Therefore, the software will display the connection of the DCS or the I/O card.

Trace a Signal in a Point-to-Point Wiring Diagram

You can follow a signal along a sequence of wires. SmartPlant Instrumentation uses wire groups, group level and group sequence data to follow a signal along a series of wires and check for any discontinuity. The wire group is the signal name representing the tag number from which the signal originates. The result is displayed in a point-to-point wiring diagram that you can print.

1. Do one of the following to open the **Point-to-Point Wiring Diagram**:
 - In the **Domain Explorer**, right-click a cable, cable set, panel, or a terminal strip and then on the shortcut menu, click **Actions > Connection**. For best results, make sure that the connection information is complete for the items that you selected.
 - In the **Domain Explorer**, right-click a cable, cable set, panel, or a terminal strip and then on the shortcut menu, click **Actions > I/O Assignment**. For best results, make sure that the I/O assignment information is complete for the items that you selected.
2. In the **Connection** window, from the **Terminal strip** list, select the terminal strip to which you want to trace the signal path.

TIP If you are working in the **I/O Assignment** window, skip steps 2 and 3.
3. From the **Cable** list, select the cable containing the wire for which you want to trace the signal path.
4. Do one of the following:
 - In the **Connection** window, select the wire for which you want to trace the signal path.
 - In the **I/O Assignment** window, under Channel Assignment, select a channel associated with an instrument tag.
5. Click  to locate a specific terminal or channel.
6. Do one of the following to generate the **Point to Point Wiring Diagram** report:
 - On the module toolbar, click .
 - On the menu bar, click **Reports > Point to Point Wiring Diagram**.
 - In the **Connection** window, right-click the selected wire and on the shortcut menu click **Point to Point Wiring Diagram**.
7. Do one of the following when prompted to preview the report:
 - Click **Yes** to preview the **Point to Point Wiring Diagram**.
 - Click **No** to print the report without displaying it on your computer.

NOTES

- To view the point-to-point diagram legend, scroll down to the lower part of the preview.
- To hide the default notation of level and sequence of the signals and tags, click **View > Show Level/Sequence**. To display the notation, click again.
- You can resize the fonts of the cable and cable set names in the **Point to Point Wiring Diagram**. For details, see *Change the Font Size of Cable and Cable Set Names* (on page 103).

Change the Font Size of Cable and Cable Set Names

1. In the **Point-to-Point Wiring Diagram**, click **View > Font Size**.
2. Do one of the following on the **Resize Fonts** dialog box:
 - In the **Size** field, type the required font size of the cable and cable set names.
 - Use the spinner to increase or decrease the font size.
3. Click **OK** to close the **Resize Fonts** dialog box and return to the **Point to Point Wiring Diagram** print preview where you see the changes you made to the cable and cable set name fonts.

Generate a Connection Report from the Connection Window

While in the **Connection** window, you can generate a connection report for the current terminal strip. The report lists all the cables connected to the selected terminal strip and shows all the cable-sets within each cable, the wire tags, their color, and terminal number. This report does not show adjacent connections.

1. In the **Connection** window, do one of the following:
 - On the menu bar, click **Reports > Connection**.
 - Click .
2. In the **Print Preview** prompt, click **Yes** to display the report print preview or click **No** to print out the report without displaying its print preview.

SECTION 10

I/O Assignment

The I/O assignment in SmartPlant Instrumentation enables you to assign a channel to a specific instrument or a fieldbus segment.

I/O assignment for an instrument is performed in two stages:

1. Associating (coupling) an instrument with a control system tag.
2. Assigning the coupled pair to a channel.

I/O assignment for a segment is also performed in two stages:

1. Associating (coupling) each virtual tag with a control system tag.
2. Assigning the coupled pairs to an I/O channel.

NOTE The software displays the available instruments according to the current I/O type. You can change the I/O type by setting the appropriate I/O assignment preference. For details, see *Set I/O Assignment Instrument Tag Preferences* (on page 105).

Set I/O Assignment Instrument Tag Preferences

Prior to effecting I/O assignment, you can set your I/O assignment preferences for the I/O types of the instrument tags. These preference options enable you to determine the I/O types of the instrument tags that become available in the **I/O Assignment** window. One setting allows you to make all instrument tags be available in the **I/O Assignment** window regardless of their I/O type. The other setting limits the availability of instrument tags only to those instruments whose I/O type is identical to the I/O type of the current I/O card.

Note that after you effect an I/O assignment and your preference was set to the **Same I/O type for tag and I/O card**, you can still change the I/O type of the I/O card if you need to do so. The software does not verify the compatibility of a changed I/O type with instrument tags already associated with I/O cards.

1. Click **File > Preferences**.
2. Expand the **Wiring** hierarchy in the tree view and select **I/O Assignment**.
3. Under **I/O assignment validation** select one of the following options:
 - **Same I/O type for tag and I/O card** — limits the I/O type of the instrument tag to be the same as the I/O type of the I/O card selected in the **I/O Assignment** window. Note that the software validates I/O type compatibility also upon connection.
 - **All I/O types** — allows the selection of all instrument tag I/O types in the **I/O Assignment** window and upon connection.
4. Click **OK**.

Effect I/O Assignment for an Instrument

I/O assignment is essentially assigning a control system tag to a free channel. I/O assignment for an instrument is performed in two stages. First you associate (couple) an instrument with a control system tag and then assign the coupled pair to a channel.

1. Press F7 to open the **Domain Explorer**.
2. In the **Domain Explorer**, do one of the following:
 - Select a parent wiring item that contains one or more I/O card child items and whose I/O type is **Fieldbus FF**.
 - Select an I/O card or an I/O termination whose I/O type is **Fieldbus FF**.
3. Right-click the selected items and then on the shortcut menu, click **Actions > I/O Assignment**.

TIPS

- The software allows you to open the **I/O Assignment** window without selecting anything in the **Domain Explorer**. On the **Wiring Module** window toolbar, click .
 - You can also open the **I/O Assignment** window for an I/O card from the **Connection** window. On the **Connection** menu, click **I/O Assignment**.
4. On the **I/O Assignment Type** dialog box, select **I/O Assignment**.
TIP For fieldbus I/O assignment, select **Segment I/O assignment**. For details, see *Configure a Fieldbus Apparatus* (on page 197).
 5. If you did not select a panel or an I/O card in the **Domain Explorer** before opening the **I/O Assignment** window, do one of the following in the **I/O Assignment** window:
 - Click  to open the **Find Channel** dialog box, where you find and select channels to display in the **I/O Assignment** window.
 - In the **I/O card details** pane, click  beside the I/O termination list to open the **Domain Explorer** where you select the required I/O cards, then click **OK** to add them to the I/O card list.
 6. From the **I/O termination** list, select an item for which you want to effect I/O assignment.
TIP The I/O type of the selected I/O card or I/O termination determines the I/O type of the initial instrument tag displayed in the **I/O type** list. You can select a different I/O type from this list if required. The list of instruments displayed in the **Tag list** data grid depends on the I/O type that you select from the **I/O type** list. If the I/O type selection is **MIXED**, instruments of all I/O types are shown. If any another I/O type is selected, only the instruments of the selected I/O type are shown.
 7. If required, select a different I/O type, which enables you to use an instrument tag for the I/O assignment whose I/O type is different from the I/O card I/O type. To do this, do one of the following:
 - If the I/O type selection is **MIXED**, select the required I/O type from the **I/O type** list.
 - If the I/O type selection is **not MIXED**, change the definitions to make the **I/O type** list available at all times by selecting the **All I/O types** preference on the **Preferences (Wiring – I/O Assignment)** dialog box. For details, see *Set I/O Assignment Instrument Tag Preferences* (on page 105).
 8. Drag an instrument or a control system tag to a free channel in the **Channel assignment** pane to effect I/O assignment.

NOTE Several assignment options are possible. For details, see the **Assign an Instrument to a Channel** section in *Effecting I/O Assignment Common Tasks*.

Effect I/O Assignment for a Segment

I/O assignment for a segment is performed in two stages. First you associate (couple) each virtual tag with a control system tag and then assign the coupled pairs to an I/O channel.

1. Press F7 to open the **Domain Explorer**.
2. In the **Domain Explorer**, do one of the following:
 - Select a parent wiring item that contains one or more I/O card child items and whose I/O type is **Fieldbus FF**.
 - Select an I/O card or an I/O termination whose I/O type is **Fieldbus FF**.
3. Right-click the selected items and then on the shortcut menu, click **Actions > I/O Assignment**.

TIP You can also open the **Segment I/O Assignment** window from the **Connection** window. On the **Connection** menu, click **I/O Assignment**.
4. On the **I/O Assignment Type** dialog box, select **I/O Assignment**.

TIP For fieldbus I/O assignment, select **Segment I/O assignment**. For details, see *Configure a Fieldbus Apparatus* (on page 197).
5. If the **I/O Assignment Type** dialog box opens, select **Segment I/O Assignment I/O assignment**.
6. From the **I/O termination** list, select an item for which you want to effect I/O assignment.
7. From the **Segment coupling** list, select a required segment type. This selection determines which items are displayed under **Segments**.
8. Drag a segment to a free channel in the **Channel assignment** pane to effect I/O assignment.

NOTE Several assignment options are possible. For details, see the **Assign an Instrument to a Channel** section in *Effecting I/O Assignment Common Tasks*.

Assign an Instrument to a Channel

The essence of I/O assignment is assigning an instrument to a channel. SmartPlant Instrumentation provides several options for this purpose. You can choose one of the following options for I/O assignment:

Select an instrument tag coupled with a control system tag and assign both of them to a free channel.

- Couple an instrument tag with an existing control system tag and then assign them to a free channel.
 - Assign an uncoupled control system tag to a free channel.
 - Assign an uncoupled instrument tag to a free channel. You will have to create a new control system tag in the process and the coupling will be done automatically.
 - Assign an uncoupled instrument tag to a channel to which a control system tag has already been assigned.
1. Open the **I/O Assignment** window.
 2. In the **Tag coupling** data grid, select one of the following:
 - An instrument tag coupled with a control system tag.
 - An instrument not coupled with a control system tag.
 - A control system tag not coupled with an instrument.
 3. Drag the selected item to a free channel in the **Channel assignment** pane.
 4. If you selected an instrument not coupled with a control system tag, do the following:

5. On the **Control System Tag** dialog box, type the name of the control system tag that SmartPlant Instrumentation creates automatically.
6. Click **OK** on the **Control System Tag** dialog box to complete the I/O assignment.

NOTE You can shift the assignment of an instrument to another unassigned channel by simply pressing and holding the Ctrl key while dragging the instrument to another unassigned channel. You cannot change the assignment this way if the target channel has already been assigned to another instrument. Note that when changing the assignment, the software retains the existing control system data.

Assign a Segment to a Channel

This option enables to assign a segment to a channel so that you can effect I/O assignment for fieldbus segments.

1. Press F7 to open the **Domain Explorer**.
2. In the **Domain Explorer**, do one of the following:
 - Select a parent wiring item that contains one or more I/O card child items and whose I/O type is **Fieldbus FF**.
 - Select an I/O card or an I/O termination whose I/O type is **Fieldbus FF**.
3. Right-click the selected items and then on the shortcut menu, click **Actions > I/O Assignment**.
4. From the **Segments** list, select an appropriate segment.

TIP The software displays only those segments that contain instruments for which virtual tags have been created.

5. Drag the selected segment to a free channel in the **Channel assignment** pane.

NOTE You can shift the assignment of a segment to another unassigned channel by simply pressing and holding the Ctrl key while dragging the segment to another unassigned channel. You cannot change the assignment this way if the target channel has already been assigned to another segment. Note that when changing the assignment, the software retains the existing control system data.

Create a Channel

This option allows you to create additional channels to the ones that the software created in a particular I/O card or terminal strip.

1. In the **Domain Explorer**, select an I/O card, I/O termination, or a terminal strip with channels.
2. Right-click the item to which you want to add a channel and then on the shortcut menu, click **New > Channel**.
3. On the **Channel Properties** dialog box, on the **General** tab, type a channel name, description and sequence as you require.
4. If you are adding a channel on a terminal strip under an I/O card or I/O termination, click the **I/O Properties** tab.
5. On the **I/O Properties** tab, select the channel type, I/O type, and signal type.
6. Type a value in the **Address** field as you require.
7. Type a minimum and maximum range and select a unit of measure for the range values.
8. Select the **Enable for use with external DCS interfaces (DeltaV)** check box to make these channel property values available for use with a DCS interface such as DeltaV.
9. Click the **Associate Symbols** tab to associate a symbol (.sym) file with the new channel. For details, see *Associate a Symbol with an Item* (on page 23).

10. Click **OK**.

Create a Control System Tag

This option allows you to create a new control system tag. Note that control system tag names must be unique. If you are using a DCS, the uniqueness applies per <plant>. If you are using a PLC, the uniqueness applies per cabinet or panel.

1. On the **I/O Assignment** window menu bar, click **Actions > New Control System Tag**.
2. On the **Control System Tag Properties (New)** dialog box, type the name of the new control system tag and click **OK**.

TIP You can now add an additional control system point configuration if needed.

3. To document additional DCS/PLC data, on the SmartPlant Instrumentation toolbar click .

TIP The **Browser Manager** opens. If you have a predefined Wiring default browser and a default view, the Browser View opens by default.

4. Do the following in the **Browser Manager**:
 5. Expand the **Control System** browser group by clicking  next to its icon or double-clicking the icon.
 6. To view the control system tag data, select the **Control System Tag Browser** and click  to create a browser view for this browser.
 7. Define the view style, filter, and sorting sequence as you require.
 8. Click  to view the selected field data in the browser view.
 9. If desired, you can edit the displayed data, sort, change the sequence of, and resize the columns. For more information on how to work with browser views, see Browser Views Common Tasks.

Create a Control System Tag in the Instrument Index Standard Browser View

1. In the **Instrument Index Module** window, do one of the following to open the Instrument Index Standard Browser view:
 - On the menu bar, click **Actions > Standard Browser View**.
 - On the toolbar, click .
2. Select the desired tag numbers.
3. Do one of the following:
 - Right-click, and on the shortcut menu, click **Apply Profile > Create Control System Tag**.
 - On the **Actions** menu, click **Apply Profile > Create Control System Tag**.
4. In the **Results** dialog box, view the CS tag creation results, and then click **Close**.

NOTE To make this feature work, you must select the **Create wiring** and **Control system** check boxes on the **Wiring and Control System** tab in the instrument type profile of the selected tag number before you attempt to create a new control system tag. For more details, see Define an Instrument Type

Rename a Control System Tag

You can change the name of an existing control system tag, for example, if you replace an existing one with a newer model.

1. In the **I/O Assignment** window, select the control system tag that you want to rename.
TIP You can select the required control system tag in the **Tag coupling** data window or in the **Channel assignment** pane.
2. Do one of the following:
 - Click **Actions > Control System Tag Properties**.
 - Click .
3. On the **Control System Tag** dialog box, type the new name of the control system tag.
4. Click **OK**.

Couple an Instrument with a Control System Tag

One of the ways to effect I/O assignment is to assign a control system tag coupled with an instrument tag to a free channel.

1. In the **I/O Assignment** window, select a free instrument in the **Tag Coupling** data grid.
2. Click **Actions > Couple Tags**.
3. Select the control system tag you want to couple with the instrument tag you have selected.
4. Click **OK**.

NOTE The instrument tag and the control system tag appear coupled in the **Tag coupling** data grid.

Decouple an Instrument and a Control System Tag

This procedure explains how to decouple an instrument tag and a control system tag where the instrument is no longer connected.

1. Open the **I/O Assignment** window.
2. In the **Tag coupling** pane, select a coupled instrument tag and control system tag.
3. Do one of the following:
 - Click **Actions > Decouple Tag No. from SC Tag**.
 - Click .

NOTE The instrument tag and the control system tag appear separately in the **Tag coupling** data grid.

Delete a Control System Tag

1. In the **I/O Assignment** window, select the control system tag that you want to delete.

TIP You can select the required control system tag in the **Tag coupling** or in the **Channel assignment** pane. If, in the **Channel assignment** pane, you select a CS tag in the that is still coupled with an instrument, the software informs you that the I/O assignment for that channel will be canceled.
2. Do one of the following:
 - Click .
 - Click **Actions > Delete Control System Tag**.
3. Click **Yes** to confirm the deletion.

Filter the Display of Data in the I/O Assignment Window

This feature enables you to filter the data displayed in the **I/O Assignment** window. You can instruct the software to display only assigned or unassigned instruments. Furthermore, you can display only uncoupled instruments, uncoupled CS tags, or both. Also, you can select a predefined Browser filter for panels.

1. In the **I/O Assignment** window, do one of the following to open the **I/O Assignment Filter** dialog box:
 - Click .
 - Click **Actions > Filter**.
2. Do one of the following:
 - From the **Browser filter for instruments** list, select a predefined filter for panels. Note that if you select a predefined filter from this list, it overrides all the other settings in this dialog box. If you do not have a predefined filter, open the **Browser Manager** and define a filter for an appropriate view that contains panels. For details, see Browser View Filter Settings.
 - Define your filter using the options in the **Filter for** and **Definition** group boxes.
3. Under **Filter for**, select the filter option to determine which instrument tags and control system tags will be displayed in the **I/O Assignment** window. Select one of the following:
 - **Unassigned tags coupled with control system tags** — display the instruments that have not yet been assigned to channels but have already been coupled with control system tags. Then, you can determine whether to display uncoupled CS tags, uncoupled instruments, or both.
 - **Assigned tags coupled with control system tags** — display only the assigned instruments that have already been coupled with control system tags. Selecting this option disables the other three options.
4. To further filter the display of unassigned instruments, uncoupled instruments, or both, under **Definition**, define an appropriate filter expression. For details, see Define a View Filter.
5. To use the current filter settings in subsequent sessions of the **I/O Assignment** window, select the **Apply filter settings** in all sessions check box.
6. To display only the instruments that were retrieved from SmartPlant Electrical, select the **SmartPlant Electrical tags** check box.
7. To limit the display of data to the current lowest plant hierarchy level, select the **Display current unit data** only check box.
8. Click **OK**.

Cancel I/O Assignment

Canceling I/O assignment involves a simple procedure of removing the instrument assignment (un-assigning) an instrument from a channel.

1. In the **I/O Assignment** window, select a channel in the **Channel assignment** pane.
2. Do one of the following:
 - Click .
 - Click **Actions > Un-assign from channel**.

NOTE The selected channel becomes vacant as the instrument and its control system tag move to the **Tag list** pane.

Generate an I/O Tag Assignment Report

This option enables you to generate a report showing I/O assignments for an I/O card or I/O termination that you select in the **I/O Assignment** window.

1. In the **I/O Assignment** window, under **I/O assignment**, select an I/O card or an I/O termination from the list.
2. Select the required I/O card from the **I/O card name** list.
3. Do one of the following:
 - Click .
 - Click **Reports > Tag assignment Report**.
4. Click **Yes** to open the print preview of the report or click **No** to send the report to your default printer.

Control System Tag Propagation

Propagation of a signal along wires may vary according to whether a field device exists and whether a tag number was assigned to an I/O channel. If the tag number was assigned to an I/O device the cables connected to the I/O terminals are also considered during the signal propagation. The method of propagating sequence numbers (within a level) used by SmartPlant Instrumentation depends on whether a field device is associated with that tag number, and whether the wiring routing is complete (there is a signal continuity between the field device and the I/O channel).

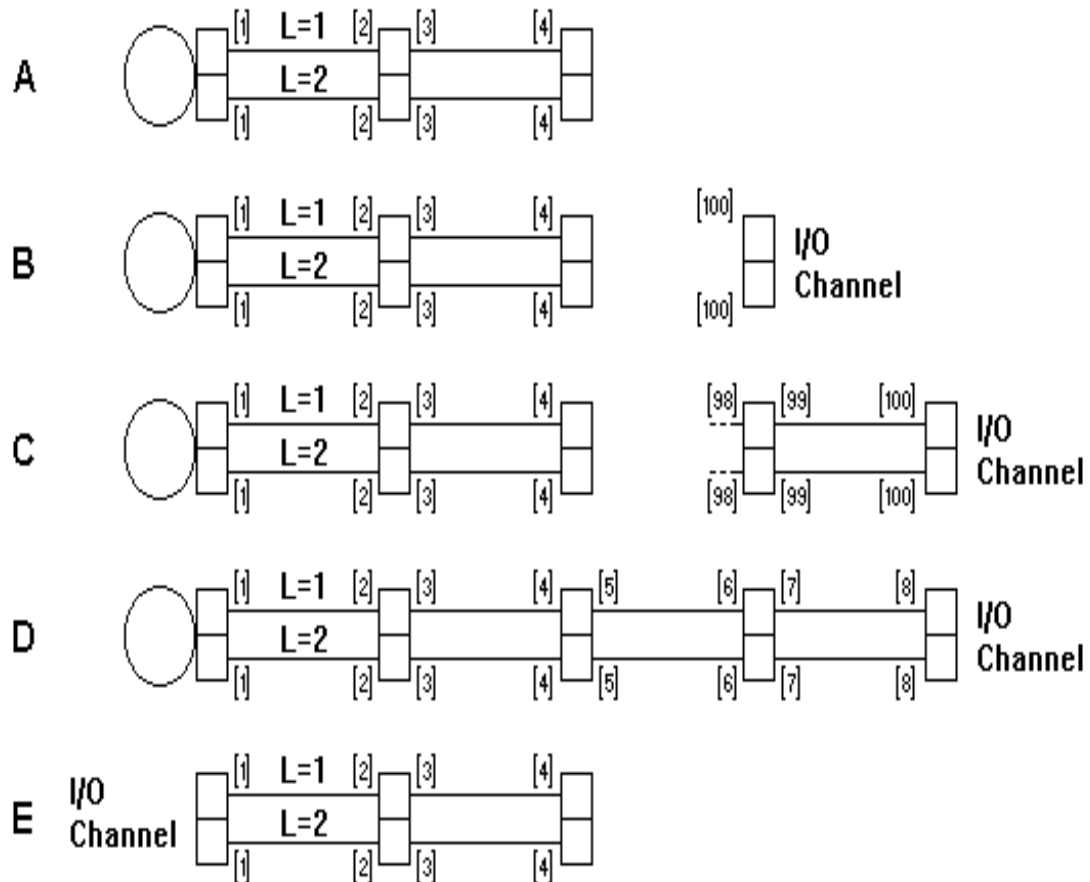


Diagram key:

- A** — Field device without a channel.
- B** — Field device and Control System channel assigned without cables connected to the Control System channel.
- C** — Field device and Control System channel assigned with cables. Wiring routing is not completed.
- D** — Field device and Control System channel assigned. Wiring routing is completed.
- E** — Control System channel assigned without a field device.

View and Edit Channel Data

This option enables you to view the channels of a selected DCS/PLC panel or all DCS/PLC panels in the current <plant>. You can also modify the control system tag names if required.

IMPORTANT If you are editing items that must be compatible with Emerson DeltaV, see Crucial Fields for the DeltaV Interface.

1. Do one of the following in the **I/O Assignment** window:
 - Click **Actions > Show I/O Data**.
 - Click .
2. To rename a control system tag, do the following:
 - a. Select the required row and click **Edit CS Tag**.
 - b. On the **Control System Tag** dialog box, type a new name as needed and click **OK**.
3. To filter the display of data on the **I/O Data** dialog box, do the following:
 - a. Click **Filter** to open the **Filter I/O Data** dialog box.
 - b. In any data field, type the value according which to filter the channels on the **I/O Data** dialog box.

Note that you can include any letter or number combination. You can include letter or number combinations, as well as wildcards. For example, an underscore (_) can substitute a single character and percent (%) can substitute multiple characters. The search is not case-sensitive.
 - c. From the drop-down lists, select a value according which to filter the channels on the **I/O Data** dialog box.
 - d. In the **Show channels** group box, choose a filtering option to determine which channels to display on the **I/O Data** dialog box.
 - e. Select the **Save** check box to save the criteria specified for the next time you open the **I/O Data** dialog box.
 - f. Click **OK** to accept your filter settings and return to the **I/O Data** dialog box.
4. To sort the displayed channels on the **I/O Data** dialog box, do the following:
 - a. Click **Sort** to open the Sort I/O Data dialog box.
 - b. In the **Column Name** list, select the column according which the channels displayed on the **I/O Data** dialog box will be sorted.
 - c. Select **Ascending** to sort the channels displayed on the **I/O Data** dialog box in ascending order; clear the check box to sort the channels in descending order.
 - d. To add a new sorting parameter, select an existing sorting parameter after which you want to add the new sorting parameter and then click **Insert**.
 - e. To delete a sorting parameter, select the required sorting parameter and click **Delete**.
 - f. Click **Clear** to discard all the sorting settings.
 - g. Select the **Save** check box if you want to save the current settings for the next time you open the **I/O Data** dialog box. Note that deleting or clearing parameters does not affect the sorting parameters that you have previously saved (by selecting the **Save** check box).
5. Click **OK** to accept your settings and return to the **I/O Data** dialog box.
6. To print out the data on the **I/O Data** dialog box, click **Report**.
7. Click **Close** to return to the **I/O Assignment** window.

NOTE If you open the **I/O Assignment** window without selecting anything in the **Domain Explorer**, the **I/O Data** dialog box shows the control system tags of the DCS/PLC cabinet you selected the last time you opened the **I/O Assignment** window.

Find I/O Channels for Assignment

This option enables you to search DCS or PLC panels in the current <plant> for I/O cards and individual channels according to required criteria. SmartPlant Instrumentation then displays these channels in the **I/O Assignment** window, and you can use them to effect an I/O assignment.

1. In the **I/O Assignment** window, do one of the following to open the **Find Channels** dialog box:
 - Click **Actions > Find Channels**.
 - Click .
2. Under **Search parameters**, type values according to which SmartPlant Instrumentation displays channels in the **Search results** data window. If you leave a field blank, it will not limit the search. Leave all fields blank in order to display all channels.
 - **TIP** You can include letter or number combinations, as well as wildcards. For example, an underscore (_) can substitute a single character, and percent (%) can substitute multiple characters. The search is not case-sensitive.
3. Under **Show channels**, select one of the following search options:
 - **Spares only** — show only spare channels in the **Search results** data window.
 - **Exclude spares** — do not display any spare channels in the **Search results** data window.
 - **All** — show all spare and assigned channels in the **Search results** data window.
4. To save the current search parameters as the channel search default, select the **Save** check box.
5. Click **Find** to search using the current settings.
6. In the **Search results** data window, select the I/O cards that you want to be available in the **I/O Assignment** window, and click **OK**.
 - **TIP** If you select a channel rather than an I/O card, SmartPlant Instrumentation displays the card to which the channel belongs in the **I/O Assignment** window.
7. In the **I/O Assignment** window, under **I/O card** name, select the required card. The software displays its channels in the **Assignment details** data window.

SECTION 11

Cross Wiring

Cross wiring in SmartPlant Instrumentation allows you to connect two terminal strips by using a cross-wiring cable. You can cross wire two terminal strips that belong to two different panels or the same panel.

The following cross-wiring options are available:

- **Manual Cross Wiring** — This is the default cross wiring mode, where you select the terminals on the strips to be cross-wired.
- **Semiautomatic Cross Wiring** — SmartPlant Instrumentation automatically finds potential targets for the signal that it will propagate. You can accept the suggested terminals and carry out the cross wiring or select different terminals to be cross-wired.
- **Automatic Cross Wiring** — SmartPlant Instrumentation automatically finds matching signals and performs the cross wiring for you.

Perform Manual Cross Wiring

Manual cross wiring is the default mode for cross wiring. In this mode, you select the terminals that you want to cross wire, select the required cross-wiring cable and then carry out the cross wiring.

1. Press F7 open the **Domain Explorer** and select one or more terminal strips for potential cross wiring. You can also select a panel or several panels if required.
2. Right-click the items you selected and then on the shortcut menu, click **Actions > Cross Wiring**.
3. In the **Cross Wiring** window, select the required terminal strip from the **Primary terminal strip** list.

TIP To add terminal strips that do not appear on the list, click  beside the **Primary terminal strip** list arrow, select the required terminal strips in the **Domain Explorer**, then click **OK** to add them to the **Primary terminal strip** list.

4. Under **Primary connection side**, click **Left** or **Right** to select the appropriate side of the secondary terminal strip that you want to connect.

TIP This option is not available if the current secondary terminal strip is a terminal strip with channels (belonging to a DCS or PLC panel). In this case, you can connect the left side only.

5. From the **Secondary terminal strip** list, select the required terminal strip that you want to cross wire.

TIP To cross wire two terminal strips belonging to different panels where you did not select one of the panels in **Cross Wiring**, click  beside the **Secondary terminal strip** list arrow, select the required terminal strip in the **Domain Explorer** and click **OK** to add it to the **Secondary terminal strip** list. Then, select that terminal strip from the **Secondary terminal strip** list for cross wiring.

6. Under **Secondary connection side**, click **Left** or **Right** to select the appropriate side of the secondary terminal strip that you want to connect.

TIP This option is not available if the current secondary terminal strip is a terminal strip with channels (belonging to a DCS or PLC panel). In this case, you can connect the left side only.

7. From the **Cross-wiring cable** list, select one of the following to define a cross-wiring cable:

- Select **CROSS WIRE** to cross wire the terminals with wires that the software creates during the cross-wiring process.
 - Select the required cross- wiring cable. If the cross wiring cable that you need is not on this list, click  beside the list arrow to open the **Wiring Explorer**. Select a cable and click **OK** to add it to the list. Note that you can also create a new cross wiring cable if the one you need does not exist in the **Domain Explorer**.
8. Select the **Prompt for cross operation message** check box to be prompted by SmartPlant Instrumentation to confirm the cross wiring. If you do not select this check box, cross wiring of the selected terminals proceeds without messages and SmartPlant Instrumentation propagates all the existing signals.
 9. In the **Primary Terminal Strip** pane, click the required terminal to be cross-wired.
 10. In the **Secondary Terminal Strip** pane, click the required terminal to be cross- wired.
 11. If you selected the **Prompt for cross operation message** check box, click **Yes** to confirm the cross wiring.

NOTES

- For an example and information about the conventions used to represent various items and connections in the **Cross Wiring** window, click the Help icon on the main toolbar and then click the required link at the bottom of the help page.
- In the **Primary Terminal Strip** pane, select the required cross-wired terminal, and then click  to open the **Point-to-Point Wiring Diagram**.
- Click  to open the **Terminal Connection** window. You can also double click  to get the same result.
- Click  to remove the cross wiring between two terminals after selecting the cross- wired terminals (shown connected by a solid black line).
- Click  to generate a report that shows all the wiring connections of the primary strip on both sides.

Perform Semiautomatic Cross Wiring

Semiautomatic cross wiring enables you to find matching signals between two terminal strips (primary and secondary). That is, SmartPlant Instrumentation can find for you a potential target for the signal that the software propagates after you perform cross wiring. You can then use this potential target to quickly effect the required cross wiring. The terminal that SmartPlant Instrumentation finds for you in the secondary terminal strip is a recommendation only and you are free to accept it or select a different terminal, as you require.

1. Press F7 open the **Domain Explorer** and select one or more terminal strips for potential cross wiring. You can also select a panel or several panels if required.
2. Do one of the following to open the **Cross Wiring** window:
 - Click **Actions > Cross Wiring**.
 - Click 
3. In the **Cross Wiring** window, select the required terminal strip from the **Primary terminal strip** list.

TIP To add terminal strips that do not appear on the list, click  beside the **Primary terminal strip** list arrow, select the required terminal strips in the **Domain Explorer**, then click **OK** to add them to the **Primary terminal strip** list.

4. From the **Secondary terminal strip** list, select the required terminal strip that you want to cross wire.

TIP To cross wire two terminal strips belonging to different panels where you did not select one of the panels in **Domain Explorer**, click  beside the **Secondary terminal strip** list arrow, select the required terminal strip in the **Domain Explorer** and click **OK** to add it to the **Secondary terminal strip** list. Then, select that terminal strip from the **Secondary terminal strip** list for cross wiring.

5. Under **Secondary connection side**, click **Left** or **Right** to select the appropriate side of the secondary terminal strip that you want to connect.

TIP This option is not available if the current secondary terminal strip is a terminal strip with channels (belonging to a DCS or PLC panel). In this case, you can connect the left side only.

6. From the **Cross-wiring cable** list, select one of the following to define a cross-wiring cable:
 - Select **CROSS WIRE** to cross wire the terminals with wires that the software creates during the cross-wiring process.
 - Select the required cross- wiring cable. If the cross wiring cable that you need is not on this list, click  beside the list arrow to open the **Domain Explorer**. Select a cable and click **OK** to add it to the list. Note that you can also create a new cross wiring cable if the one you need does not exist in the **Domain Explorer**.
7. Select the **Prompt for cross operation message** check box to be prompted by SmartPlant Instrumentation to confirm the cross wiring. If you do not select this check box, cross wiring of the selected terminals proceeds without messages and SmartPlant Instrumentation propagates all the existing signals.
8. In the **Primary Terminal Strip** pane, click the required terminal to be cross-wired.
9. Do one of the following to find a matching target signal for potential cross wiring:
 - Click **View > Target Signal**.
 - Click . SmartPlant Instrumentation searches for a matching signal in the current <plant> and adds the terminals that it finds in the **Secondary Terminal Strip** pane. SmartPlant Instrumentation indicates each of the potential terminals by placing an arrow  beside it in the **Secondary Terminal Strip** pane.
10. Select the required terminal in the **Secondary Terminal Strip** pane to cross wire with the selected terminal in the primary terminal strip.

NOTES

- For an example and information about the conventions used to represent various items and connections in the **Cross Wiring** window, click the Help icon on the main toolbar and then click the required link at the bottom of the help page.
- In the **Primary Terminal Strip** pane, select the required cross-wired terminal, and then click  to open the **Point-to-Point Wiring Diagram**.
- Click  to open the **Terminal Connection** window. You can also double click  to get the same result.
- Click  to remove the cross wiring between two terminals after selecting the cross- wired terminals (shown connected by a solid black line).
- Click  to generate a report that shows all the wiring connections of the primary strip on both sides.

Set Cross Wiring Preferences

This procedure explains how to create a new cross-wiring cable and use it in your current cross wiring operation. If you do not want to create your own cross-wiring cables and let SmartPlant Instrumentation create a cable for you, in the **Cross Wiring** window, select **CROSS WIRE** from the **Cross-wiring cable** list.

1. Click **File > Preferences**.
2. Expand the **Wiring** hierarchy in the tree view and select **Cross Wiring**.
3. Under **Automatic cross wiring**, select one of the following options:
 - **Different strips, same panel** — Automatically cross-wires two signals between two different strips belonging to the same panel (this is the default selection).
 - **Different panels (no DCS or PLC panels)** — Automatically cross-wires signals between two strips belonging to two different panels in the current plant, except for strips with channels (that is, I/O cards).
 - **Different panels including DCS or PLC panels (terminal sequence connection)** — Automatically cross-wires signals between two strips belonging to two different panels in the current plant, including strips with channels where the connection will be done according to the terminal sequence that is shown in the dialog box. Note that we do not recommend using the third option unless you are absolutely certain that this option will select matching signals. In any case, do not forget to change the setting back to the default (**Different strips, same panel**) when done.
4. From the **Default wire color** list select the required wire color. Selecting a default wire color determines the cross-wire color in the **Connection** window. The wire color data in the list is retrieved from the **Wire Color** dialog box.
5. Click **OK**.

Perform Automatic Cross Wiring

Automatic cross wiring enables you to instruct SmartPlant Instrumentation to automatically find matching signals for you and cross-wire the relevant terminals in a single operation. Two signals match if:

- The signals are between two terminal strips belonging to the same panel.
- Both signals are on the same signal level.
- The first signal comes to the first terminal from a device panel (the signal sequence is less than 50) and the second signal comes from a control system (the signal sequence is greater than 50.)

You can set the default definitions for matching signals by using the **Preferences** dialog box.

IMPORTANT You can set the default definitions for matching signals by using the **Preferences** dialog box. For details, see *Set Cross Wiring Preferences* (on page 120).

1. Press F7 open the **Domain Explorer** and select one or more terminal strips for potential cross wiring. You can also select a panel or several panels if required.
2. Right-click the selected item and on the shortcut menu, click **Actions > Cross Wiring**.
3. In the **Cross Wiring** window, select the required terminal strip from the **Primary terminal strip** list.

TIP To add terminal strips that do not appear on the list, click  beside the **Primary terminal strip** list arrow, select the required terminal strips in the **Domain Explorer**, then click **OK** to add them to the **Primary terminal strip** list.

4. Under **Primary auto cross- wiring side**, click **Left** or **Right** to select the appropriate side of the primary terminal strip that you want to connect.
5. From the **Secondary terminal strip** list, select the required terminal strip that you want to cross wire.

TIP To cross wire two terminal strips belonging to different panels where you did not select one of the panels in **Domain Explorer**, click  beside the **Secondary terminal strip** list arrow, select the required terminal strip in the **Domain Explorer** and click **OK** to add it to the **Secondary terminal strip** list. Then, select that terminal strip from the **Secondary terminal strip** list for cross wiring.

6. Under **Secondary connection side**, click **Left** or **Right** to select the appropriate side of the secondary terminal strip that you want to connect.

TIP This option is not available if the current secondary terminal strip is a terminal strip with channels (belonging to a DCS or PLC panel). In this case, you can connect the left side only.
7. From the **Cross-wiring cable** list, select one of the following to define a cross-wiring cable:
 - Select **CROSS WIRE** to cross wire the terminals with wires that the software creates during the cross-wiring process.
 - Select the required cross- wiring cable. If the cross wiring cable that you need is not on this list, click  beside the list arrow to open the **Domain Explorer**. Select a cable and click **OK** to add it to the list. Note that you can also create a new cross wiring cable if the one you need does not exist in the **Domain Explorer**.
8. Select the **Prompt for cross operation message** check box to be prompted by SmartPlant Instrumentation to confirm the cross wiring. If you do not select this check box, cross wiring of the selected terminals proceeds without messages and SmartPlant Instrumentation propagates all the existing signals.
9. Do one of the following to find matching signals:
 - Click **Actions > Auto**
 - Click . SmartPlant Instrumentation finds and selects the matching signals and the terminals that it can cross- wire automatically. The **Secondary Terminal Strip** pane does not appear in the **Cross Wiring** window.
10. Do one of the following:
 - Click  to cross wire the selected terminals. SmartPlant Instrumentation replaces the selection with a solid black line indicating that the cross wiring has been done.
 - Click  to discard the selection made by the software and return to manual mode.
11. After the software cross wired the terminals, click  to return to manual mode.

NOTES

- For an example and information about the conventions used to represent various items and connections in the **Cross Wiring** window, click the Help icon on the main toolbar and then click the required link at the bottom of the help page.

- In the **Primary Terminal Strip** pane, select the required cross-wired terminal, and then click  to open the **Point-to-Point Wiring Diagram**.
- Click  to open the **Terminal Connection** window. You can also double click  to get the same result.
- Click  to remove the cross wiring between two terminals after selecting the cross-wired terminals (shown connected by a solid black line).
- Click  to generate a report that shows all the wiring connections of the primary strip on both sides.

Create a Cross Wiring Cable

1. In the **Cross Wiring** window, click  beside the **Cross-wiring cable** list arrow to open the **Domain Explorer**.
2. Right-click the **Cross Cables** folder, and on the shortcut menu, click **New > Cross Cable**.
3. In the **Cable Properties** dialog box, enter the required values and then click **OK**.
4. Select the new cable in the **Domain Explorer** and click **OK** to add this cable to the **Cross-wiring cable** list.
5. Select the new cable in the **Cross-wiring cable** list to be used in the cross wiring operation.

Generate a Panel Strip Report from the Cross Wiring Window

This option enables you to generate a panel - strip report that displays the connections and adjacent connections for the terminal strip you selected from the **Primary terminal strip** list in the **Cross Wiring** window.

1. Do one of the following in the **Cross Wiring** window:
 - Click **Reports > Panel- Strip**.
 - Click  on the module toolbar.
2. Click **Yes** to open the print preview or click **No** to print the report without displaying it on your screen.

NOTE A Panel-Strip report (with adjacent connections, no style), cannot be printed on a portrait orientated A4 page.

SECTION 12

Auto-Wiring

Auto-wiring allows you to make automatic batch connections between specified terminal strips. This feature facilitates faster and more efficient wiring design especially when working on a grass-root project where the required wiring items already exist and you need to connect them. The essence of this feature is that you create an auto-wiring routing task where you define a cable that will connect two designated panels.

You can define an auto-wiring task for two different kinds of connection:

- A connection between two existing panels.
- A connection between a panel and a control system that will create cross wires or cross cables.

There are two prerequisites for auto-wiring routing tasks:

- The required panels and terminal strips must already exist.
- The required reference cables and connection types must be defined and ready for use.

Furthermore, for cross wiring auto-wiring routing tasks, you must also ensure the following before you start defining your task:

- The required instrument tags and their device panels already exist and they are connected to their junction boxes.
- There are instrument tags with pre-assigned I/O points – essential for cross-wiring tasks.
- There are defined reference wiring profiles for the required instrument types.

NOTES

- For details about various auto-wiring tasks, see *Defining and Executing Auto-Wiring Tasks* (on page 123).
- You cannot create an auto-wiring task for panels that are defined as Fieldbus or Telecom panels.

Defining and Executing Auto-Wiring Tasks

An auto-wiring task is a list of commands that SmartPlant Instrumentation runs according to pre-selected parameters. You define the End 1 and End 2 connections as well as the cable that SmartPlant Instrumentation uses to make the connection. The cable that the software uses to make the connection is automatically created by the software. The definition of this cable is based on the reference cable that you select. The software checks whether a cable based on the selected reference cable exists in the **Domain Explorer** and whether this cable has a sufficient number of unconnected cable sets. If not, SmartPlant Instrumentation creates a new cable based on the selected reference cable to accommodate the required connections. You can customize the cable name as needed or accept the suggested names. The software proposes cable end names based on the names of the panels to be connected to End 1 and End 2. Any additional cables that the software creates have identical names with a numeric suffix that is incremented by one for each new cable; for example XYZ.1, XYZ.2, and so forth.

You can define the following auto-wiring tasks:

- **Connecting two junction boxes or marshaling racks** — When defining an auto-wiring task for two junction boxes, you need to define the End 1 and End 2 connections as well as the cable that SmartPlant Instrumentation uses to connect the two panels.

- **Connecting a junction box or a marshaling rack to a control system** — When defining this kind of auto-wiring task, you define the End 1 connection and the auto-wiring cable. The End 2 connection options are not available as SmartPlant Instrumentation checks for available DCS or PLC panels with matching signals. This means that you have to pre-assign the required control system tags to the tag numbers (for details, see *I/O Assignment* (on page 105)). At this point, you also need to select an appropriate cross wiring option to connect the control system. After the software executes the task, open the **Connection** window for the selected junction box and display the adjacent connection to make sure that the software has executed the auto-wiring task correctly.
- **Connecting a pre-assigned junction box to another junction box or a control system** — In this case, you need to make sure that you have carried out the required I/O assignment for the control system you are going to connect and that you pre-assigned the necessary junction box and its terminal strip to the required field device. Then, you proceed to define the End 2 connection or select to connect the pre-assigned junction box to a control system.

Note that you define and execute auto-wiring tasks in the **Auto-Wiring Browser View** window.

For detailed description of the various procedures, see *Auto-Wiring Connections Common Tasks*.

Flow of Activities for Auto-Wiring

The following is a suggested flow of auto-wiring activities. The sequence in which you perform these activities depends on your work preferences. In any case, make sure that you perform the first four steps in the following list before proceeding further.

1. Define a view profile for a new **Auto-Wiring Routing Task** browser in the **Wiring** group of the **Browser Manager**. For details, see *Define a View Profile*.
2. Add and define a new view for the **Auto-Wiring Routing Task** browser. For details, see *Add a New View*.
3. If needed, define a view profile for the **JB Pre-Assignment** browser in the **Instrument Index** group of the **Browser Manager** and then add and define a new view for the **JB Pre-Assignment** browser.
4. Pre-assign the required junction boxes to tag numbers. For details, see *Pre-Assign Junction Boxes to Device Panels* (on page 125).
5. Connect the junction boxes to the device panels using one of the following methods:
6. Use the **Device Cable Batch Connection** feature. For details, see *Connect Device Cables to a Terminal Strip in Batch Mode* (on page 95).
 - Execute the auto-wiring routing task. For details, see *Defining and Executing Auto-Wiring Tasks* (on page 123).
 - Connect the junction boxes manually.
7. Continue connecting the rest of the junction boxes.
8. Create and run a new auto-wiring routing task to connect the last junction box to the control system (or another type of panel).

Pre-Assign Junction Boxes to Device Panels

You use this feature to pre-assign specific junction boxes to selected instruments. This pre-assignment serves several purposes:

- Pre-assign the required junction boxes to define an Auto-Wiring routing task.
- Connect multiple pre-assigned device cables in batch mode.
- Filter the display of cables in the **Domain Explorer** so that only pre-assigned device cables for connection with junction boxes are displayed.

1. On the main toolbar, click  to open the **Browser Manager**.
2. Expand the **Instrument Index** browser group.

TIP You can also click  on the **Auto-Wiring Browser View** toolbar to open the **JB Pre-Assignment Browser View** window.

3. Select the **JB Pre-assignment** browser and add a new browser view to this browser.

IMPORTANT Make sure that you select the **Set as default view** check box if you want to use the new browser view for auto-wiring.

4. Click  to open the new view.
5. In the **Browser View** window, highlight the required tag number to which you want to assign a junction box.
6. Click in the **Junction Box Name** field and select an appropriate junction box that will be pre-assigned to the highlighted instrument tag.

TIP If a junction box has already been connected to a device panel, the **Junction Box Name** field displays the name of the connected junction box and you cannot rename the junction box here.

7. Click in the **Strip Name** field and select the required strip in the pre-assigned junction box.
8. Repeat steps 5 through 7 for each device panel to which you want to pre-assign a junction box.
9. Edit the data in the **Browser View** window as required and close the **Browser View** window. Save the data when prompted.

NOTES

- You can now select the appropriate junction boxes and perform batch connection of device cables in the **Batch Device Cable Connection** window.
- You can define an Auto-Wiring routing task.
- You can filter the display of device cables in the **Domain Explorer**. For details, see Define a Filter for Pre-Assigned Device Cables.

Auto-Wire Pre-Assigned Junction Boxes

This procedure explains how to define and execute an auto-wiring task for a pre-assigned junction box. Note that you must pre-assign the required junction box and make all the required preparations prior to defining and executing a task.

IMPORTANT You must pre-assign the required junction boxes and make all the required preparations prior to defining and executing a task. For details, see *Pre-Assign Junction Boxes to Device Panels* (on page 125).

1. Start the Wiring module.
2. On the menu bar, click **Actions > Auto-Wiring**.
3. In the **Browser View – New Auto-Wiring Routing Task** window, select the new task and do one of the following:
 - Click **Actions > Edit Task**.
 - Click .

TIP The **Auto-Wiring Routing Task** dialog box opens where all the **End 1 connection** properties except for the **Connection type** and **Overall shield terminal connection** have been defined. You can accept the displayed values or modify them as needed.

4. On the **Auto-Wiring Routing Task** dialog box, complete the **End 1 connection** definition as follows:
 - a. Select the required connection type from **Connection type** list.
 - b. If required, select the appropriate overall shield terminal connection.
5. In the **End 2 connection** group box, select one of the following options:
 - Select the **Control system** check box if you want to connect the pre-assigned junction box to a DCS or PLC cabinet. (For details, see *Define and Execute an Auto-Wiring Task for Control Systems* (see "Auto-Wire Control Systems" on page 128).
 - Do not select the **Control system** check box to connect the pre-assigned junction box to another junction box or marshaling rack.
6. If you did not select the **Control system** check box, define the end 2 connection of the auto-wiring cable as follows:
 - a. Click  next to the **Panel** field to select the required panel.
 - b. In the **Domain Explorer**, select a panel that contains at least one terminal strip with unconnected terminals and then click **OK**.
 - c. Select a terminal strip from the **Strip** list.
 - d. Select a terminal that will serve as the first terminal.
 - e. Select a terminal side.
 - f. Select a connection type.
 - g. If required, select an appropriate overall shield terminal connection.
 - h. Use the options in the **Cable** group box to define the cable that SmartPlant Instrumentation will use to connect the two panels:
 - i. From the **Reference cable** list, select the appropriate reference cable that SmartPlant Instrumentation will use as the source for the creation of a cable that will connect the two panels. The software duplicates the new cable in the **Domain Explorer**.
 - j. In the **Cable name** data field, accept or type the name of the cable that SmartPlant Instrumentation will create to connect the two panels. The cable names suggested by the software are composed of the End 1 and End 2 panel names. Additional cables have their names incremented by one, for example XYZ.1, XYZ.2, and so forth.

- k. Click **OK** to complete the task definition and return to the **Auto-Wiring Browser View – New Auto-Wiring Routing Task** window.
7. In the **Browser View – New Auto-Wiring Routing Task** window, click .

NOTES

- After notifying you that the task has been executed successfully, the task status changes to **Done**. For details, see *Auto-Wiring Task Statuses* (on page 131).
- You can open the **Connection** window for the selected junction box to display the adjacent connection to make sure that the auto-wiring task has been executed correctly.

Auto-Wire Two Panels

This procedure explains how to define and execute an auto-wiring task for two panels (regardless of their signals). Note that you must make all the required preparations prior to defining and executing a task.

IMPORTANT Make sure that you have made all the required preparations that are specified in *Auto-Wiring* (see "Auto-Wiring" on page 123).

1. Start the Wiring module.
2. On the menu bar, click **Actions > Auto-Wiring**.
3. In the **Browser View – New Auto-Wiring Routing Task** window, select the new task and do one of the following:
 - Click **Actions > New**.
 - Click .
4. On the **Auto-Wiring Routing Task** dialog box, define the **End 1 connection** as follows:
 - a. Click  next to the **Panel** field to select the required panel.
 - b. In the **Domain Explorer**, select a panel that contains at least one terminal strip with unconnected terminals and then click **OK**.
 - c. Select a terminal strip from the **Strip** list.
 - d. Select a terminal that will serve as the first terminal.
 - e. Select a terminal side.
 - f. Select a connection type.
 - g. If required, select an appropriate overall shield terminal connection.
5. In the **End 2 connection** group box, do the following:
 - a. Click  next to the **Panel** field to select the required panel.
 - b. In the **Domain Explorer**, select a panel that contains at least one terminal strip with unconnected terminals and then click **OK**.
 - c. Select a terminal strip from the **Strip** list.
 - d. Select a terminal that will serve as the first terminal.
 - e. Select a terminal side.
 - f. Select a connection type.
 - g. If required, select an appropriate overall shield terminal connection.
 - h. Use the options in the **Cable** group box to define the cable that SmartPlant Instrumentation will use to connect the two panels:
 - i. From the **Reference cable** list, select the appropriate reference cable that SmartPlant Instrumentation will use as the source for the creation of a cable that will connect the two panels. The software duplicates the new cable in the **Domain Explorer**.

- j. In the **Cable name** data field, accept or type the name of the cable that SmartPlant Instrumentation will create to connect the two panels. The cable names suggested by the software are composed of the End 1 and End 2 panel names. Additional cables have their names incremented by one, for example XYZ.1, XYZ.2, and so forth.
 - k. Click **OK** to complete the task definition and return to the **Auto-Wiring Browser View – New Auto-Wiring Routing Task** window.
6. In the **Browser View – New Auto-Wiring Routing Task** window, click .

NOTES

- After notifying you that the task has been executed successfully, the task status changes to **Done**. For details, see *Auto-Wiring Task Statuses* (on page 131).
- You can open the **Connection** window for the selected junction box to display the adjacent connection to make sure that the auto-wiring task has been executed correctly.

Auto-Wire Control Systems

You use this procedure to define and execute an auto-wiring task for a control system. Note that prior to defining such a task, you must assign the required control system I/O channels to specific tag numbers.

IMPORTANT

- Prior to defining such an auto-wiring task for a control system, you must assign the required control system I/O channels to specific tag numbers. For details, see *I/O Assignment* (on page 105).
 - Make sure that you have made all the required preparations that are specified in *Auto-Wiring* (see "Auto-Wiring" on page 123).
 - For auto-wiring tasks that connect pre-assigned junction boxes to control systems, see *Auto-Wire Control Systems Connected to Pre-Assigned Junction Boxes* (on page 130).
1. Start the Wiring module.
 2. On the menu bar, click **Actions > Auto-Wiring**.
 3. In the **Browser View — New Auto-Wiring Routing Task** window, select the new task and do one of the following:
 - Click **Actions > New**.
 - Click .
 4. On the **Auto-Wiring Routing Task** dialog box, define the **End 1 connection** as follows:
 - a. Click  next to the **Panel** field to select the required panel.
 - b. In the **Domain Explorer**, select a panel that contains at least one terminal strip with unconnected terminals and then click **OK**.
 - c. Select a terminal strip from the **Strip** list.
 - d. Select a terminal that will serve as the first terminal.
 - e. Select a terminal side.
 - f. Select a connection type.
 - g. If required, select an appropriate overall shield terminal connection.
 - h. In the **End 2 connection** group box, select the **Control system** check box.

TIP Selecting the **Control system** check box disables all the options in the **End 2 connection** group box (except for **Connection type**) so that SmartPlant Instrumentation can search for an available DCS or PLC panel with a matching signal.
 - i. From the **Connection type** list, select the required End 2 connection type.

5. In the **Cross wiring** group box, select an appropriate cross wiring option to connect the control system to the junction box:
 - **Single cross wires** — Connects the control system using a single wire created by SmartPlant Instrumentation automatically during the auto-wiring process.
 - **Cables created from reference** — Connects the control system using a cable that SmartPlant Instrumentation creates by duplicating the reference cable that you select from the **Reference cable** list in the **Cable** group box. The software checks whether such a cable exists in the **Domain Explorer** and if so, whether this cable is connected to the panel selected in the **End 1 connection** group box. If this cable has unconnected cable sets, the software connects them and then creates additional cables so that all the terminals are connected. Note that in this case, the software connects all the cable sets whether they are required or not. Therefore, for this option to work properly, you must create appropriate reference cables and connection types before starting auto-wiring.
 - **One cable per tag** — Connects the control system using a separate cable for each tag number. The software creates each cable by duplicating the reference cable that you select from the **Reference cable** list in the **Cable** group box. SmartPlant Instrumentation creates the required number of cables according to the number of tag signals it detects. The software names each cable according to the signal name. You can add a prefix and a suffix to the cable name by typing the required string in the **Cable name prefix** and **Cable name suffix** fields.
6. Use the options in the **Cable** group box to define the cable for connecting the pre-assigned junction box to the control system (not available if you selected the **Single cross wires** option):
 - a. From the **Reference cable** list, select an appropriate reference cable that SmartPlant Instrumentation uses as a source for the creation of a cable that will connect the two panels. The software duplicates the new cable in the **Domain Explorer**.
 - b. In the **Cable name** data field, the software suggests a cable name composed of the End 1 and End 2 panel names. Accept this name or type a name of your choice for the cable that SmartPlant Instrumentation will create to connect the two panels. Any additional cables that the software creates have identical names with a numeric suffix that is incremented by one for each new cable; for example XYZ.1, XYZ.2, and so forth.
 - c. Click **OK** to complete the task definition and return to the **Browser View — New Auto-Wiring Routing Task** window.
7. Click .

NOTES

- After notifying you that the task has been executed successfully, the task status changes to **Done**. For details, see *Auto-Wiring Task Statuses* (on page 131).
- You can open the **Connection** window for the selected junction box to display the adjacent connection to make sure that the auto-wiring task has been executed correctly.

Auto-Wire Control Systems Connected to Pre-Assigned Junction Boxes

This procedure explains how to define and execute an auto-wiring task for a control system that is to be connected to a pre-assigned junction box. Note that prior to defining such a task, you must assign the required control system to specific tag numbers.

IMPORTANT

- You must pre-assign the required junction boxes and make all the required preparations prior to defining and executing a task. For details, see *Pre-Assign Junction Boxes to Device Panels* (on page 125).
 - Assign the required control system I/O channels to specific tag numbers. For details, see *I/O Assignment* (on page 105).
 - Make sure that you have made all the required preparations that are specified in *Auto-Wiring* (see "Auto-Wiring" on page 123).
 - For auto-wiring tasks involving a connection to a junction box that has not been pre-assigned, see *Auto-Wire Control Systems* (on page 128).
1. Start the Wiring module.
 2. On the menu bar, click **Actions > Auto-Wiring**.
 3. In the **Browser View — New Auto-Wiring Routing Task** window, select the new task and do one of the following:
 - Click **Actions > Edit Task**.
 - Click .

TIP The **Auto-Wiring Routing Task** dialog box opens where all the **End 1 connection** properties except for the **Connection type** and **Overall shield terminal connection** have been defined. You can accept the displayed values or modify them as needed.

4. On the **Auto-Wiring Routing Task** dialog box, complete the **End 1 connection** definition as follows:
 - a. Select the required connection type from **Connection type** list.
 - b. If required, select the appropriate overall shield terminal connection.
 - c. In the **End 2 connection** group box, select the **Control system** check box.

TIP Selecting the **Control system** check box disables all the options in the **End 2 connection** group box (except for **Connection type**) so that SmartPlant Instrumentation can search for an available DCS or PLC panel with a matching signal.
 - d. From the **Connection type** list, select the required End 2 connection type.
5. In the **Cross wiring** group box, select an appropriate cross wiring option to connect the control system to the junction box:
 - **Single cross wires** — Connects the control system using a single wire created by SmartPlant Instrumentation automatically during the auto-wiring process.
 - **Cables created from reference** — Connects the control system using a cable that SmartPlant Instrumentation creates by duplicating the reference cable that you select from the **Reference cable** list in the **Cable** group box. The software checks whether such a cable exists in the **Domain Explorer** and if so, whether this cable is connected to the panel selected in the **End 1 connection** group box. If this cable has unconnected cable sets, the software connects them and then creates additional cables so that all the terminals are connected. Note that in this case, the software connects all the cable sets whether they are required or not. Therefore, for this option to work properly, you must create appropriate reference cables and connection types before starting auto-wiring.

- **One cable per tag** — Connects the control system using a separate cable for each tag number. The software creates each cable by duplicating the reference cable that you select from the **Reference cable** list in the **Cable** group box. SmartPlant Instrumentation creates the required number of cables according to the number of tag signals it detects. The software names each cable according to the signal name. You can add a prefix and a suffix to the cable name by typing the required string in the **Cable name prefix** and **Cable name suffix** fields.
6. Use the options in the **Cable** group box to define the cable for connecting the pre- assigned junction box to the control system (not available if you selected the **Single cross wires** option):
 - a. From the **Reference cable** list, select an appropriate reference cable that SmartPlant Instrumentation uses as a source for the creation of a cable that will connect the two panels. The software duplicates the new cable in the **Domain Explorer**.
 - b. In the **Cable name** data field, the software suggests a cable name composed of the End 1 and End 2 panel names. Accept this name or type a name of your choice for the cable that SmartPlant Instrumentation will create to connect the two panels. Any additional cables that the software creates have identical names with a numeric suffix that is incremented by one for each new cable; for example XYZ.1, XYZ.2, and so forth.
 - c. Click **OK** to complete the task definition and return to the **Browser View — New Auto-Wiring Routing Task** window.
 7. Click .

NOTES

- After notifying you that the task has been executed successfully, the task status changes to **Done**. For details, see *Auto-Wiring Task Statuses* (on page 131).
- You can open the **Connection** window for the selected junction box to display the adjacent connection to make sure that the auto-wiring task has been executed correctly.

Auto-Wiring Task Statuses

After you define an auto-wiring task, SmartPlant Instrumentation assigns it a status. Tasks can have different statuses that are set according to the task definitions. You can change certain task statuses if needed.

The following table defines the various task statuses and explains which statuses you can change.

Status	Description	Can Be Changed To
New	A newly created task that has not yet been executed.	Hold
Hold	A task that has been put on hold. This task cannot be executed until you change its status to Execute .	Execute
Execute	A task whose status has been changed from Hold . This task can be executed.	Cannot be changed
Done	A task that has already been executed. Such a task cannot be executed again.	Cannot be changed

Change an Auto-Wiring Task Status

After you define an auto-wiring task, SmartPlant Instrumentation assigns it a status. Tasks can have different statuses that are set according to the task definitions. You can change certain task statuses if needed.

1. In the **Auto-Wiring Browser View** window, right-click a task.
2. On the shortcut menu, click **Task Status**.
3. Click the option that you require.

SECTION 13

Signal Propagation

Signal propagation is a method used in SmartPlant Instrumentation to associate wires with instruments. A signal or a tag signal is a software identifier that is associated with a particular instrument tag, and is generated in a device panel or locally within other types of panels. A tag signal can also be created as a result of I/O assignment.

When you connect or disconnect a cable, cable set, or wire, SmartPlant Instrumentation automatically updates the wire tags and the signals that are carried by the wires. Tag signal propagation takes place automatically, with consequent updating of wire names, and signal sequences. Signal sequence values are sequential numbers assigned consistently to all the wires within a given wire group along the signal path of that wire group. Automatic tag signal propagation also takes place after you effect I/O assignment. For details of various tag signal propagation results, see *Possible Cases of Signal Propagation* (on page 133).

However, there may be situations in which tag signal propagation does not occur automatically. In this case, SmartPlant Instrumentation offers you two options for semiautomatic propagation of a selected tag signal. You can re-propagate a tag signal or force tag signal propagation. For details, see *Force Signal Propagation* (on page 135) and *Re-Propagate a Tag Signal* (on page 135).

Also, you can effect signal propagation at terminal strip level by manually entering or changing a signal or its level at any point along the wiring path.

TIPS

- Unlike wires, jumpers do not propagate signals. You can assign a wire group to a jumper manually on the **Terminal Connection** dialog box.
- Tag signal propagation for a shield takes place automatically without the need to connect that shield to a device panel terminal.
- Tag signal propagation stops if it reaches a point where multiple optional paths exist.
- You can re-propagate or force propagation of tag signals if this is required.
- You can intervene at any point of the wire path and change tag number propagation manually.

You can generate a point-to-point wiring diagram and view the signal path after you finish assigning an instrument tag to a channel in an I/O card. For information about point-to-point diagrams, see *Trace a Signal in a Point-to-Point Wiring Diagram* (on page 102).

Possible Cases of Signal Propagation

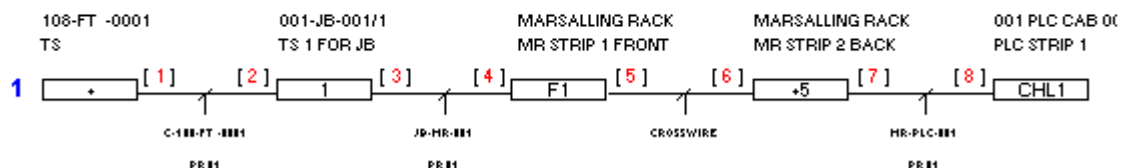
After you effect I/O assignment, the software automatically propagates the tag signals. Four situations can exist:

Case 1

In this example, the signal sequence is from the device panel up to the PLC cabinet with all the wiring:

Signal / Tag Number: 108-FT -0001

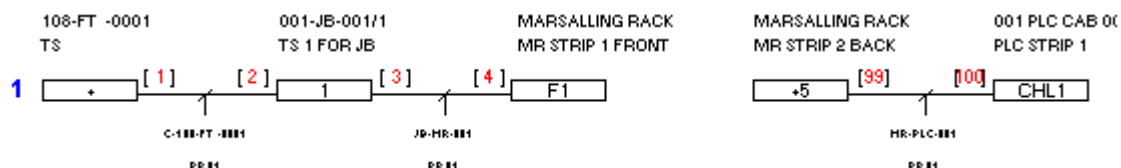
Level

**Case 2**

In this case the connection sequence starts from the device panel and continues to the PLC cabinet, however, there is a discontinuity between Strip 1 and Strip 2 of the marshaling rack:

Signal / Tag Number: 108-FT -0001

Level



Note that the wiring sequence can be different depending on the completeness of the connection:

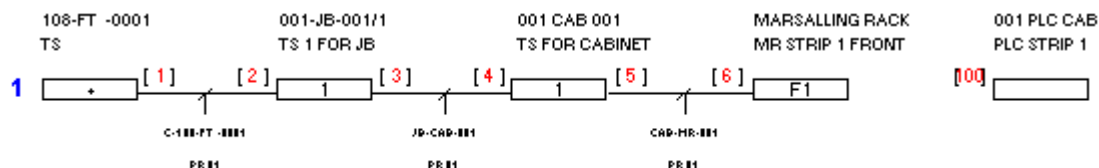
- From 1 to 8 (as in the first case) — connected from the device panel to the PLC.
- From 1 to 4 and from 99 to 100 (as in the second case) — there is a break in the middle of the connection.

Case 3

The signal sequence is from the device panel to the PLC cabinet however, there is a discontinuity between the marshaling rack and the PLC cabinet.

Signal / Tag Number: 108-FT -0001

Level

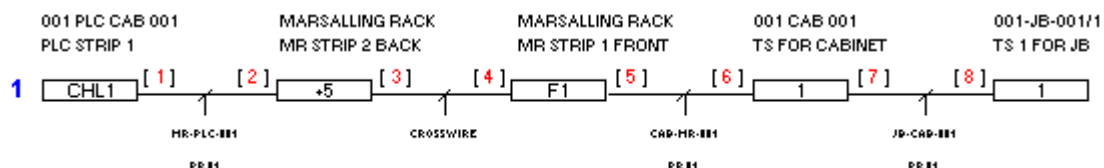


Case 4

The connection sequence starts from the PLC cabinet and continues to the junction box. The device panel is absent.

Signal / Tag Number: 108-FT -0001

Level



Re-Propagate a Tag Signal

You use this feature when the software does not fully complete the automatic propagation of a tag signal. This procedure explains how to propagate a tag signal from a selected device panel, through all the connected cables and panels, until it encounters a different signal. After it encounters a different signal, the software does not make any further changes in the propagation.

1. Press F7 to open the **Domain Explorer**.
2. Right-click a device panel from which a signal originates.
3. On the shortcut menu, click **Actions > Re-Propagate Signal**.

Force Signal Propagation

You use this feature when the software does not fully complete the propagation of a tag signal automatically. This procedure explains how to force the propagation of a tag signal from a selected device panel, through all the connected cables and panels so that the re-propagated signal overrides all the existing signals that it encounters.

1. Press F7 to open the **Domain Explorer**.
2. Right-click a device panel from which a signal originates.
3. On the shortcut menu, click **Actions > Force Signal Propagation**.

Propagate a Tag Signal Manually

During your wiring design, situations can arise in which the propagation of a tag signal does not occur automatically. SmartPlant Instrumentation lets you effect signal propagation at terminal strip level by manually entering or changing a signal or its level at any point along the wiring path. You need to propagate tag signals manually when:

- There is a split in the wiring path.
 - There are terminal strips that emulate barriers.
 - You cross-wire two terminals, one of which already receives a signal.
 - You remove a signal that is stopping tag signal propagation.
 - You add another signal level at an exit point of one of the terminal strips.
1. Do one of the following:
 - In the **Connection** window, from the **Terminal Strip** list, select the terminal strip for which you want to effect signal propagation manually.

- In the **Cross Wiring** window, from the **Primary terminal strip** list, select the terminal strip for which you want to effect signal propagation manually.
2. Double-click  next to the required wired terminal.
3. On the **Terminal Connection** dialog box, under **Signal**, click and then select from the list the signal that you want to propagate.
4. Under **Signal Level**, click and select from the list, a value for the signal level.
TIP You can also select zero to indicate that a signal does not have a signal level setting. This means that all the wires in the current wire group are propagated on the same level.
5. Under **Sequence**, click and select from the list a connection sequence.
6. Click **Save** to propagate the signal.
7. To navigate to other terminals for which you need to change propagation settings, do one of the following:
 - Click **Next** or **Previous**.
 - Under **Terminal number**, select a terminal that you require.

NOTES

- Repeat the above procedure for each terminal strip that requires manual signal propagation.
- If you remove a signal at any point of the wiring path, SmartPlant Instrumentation stops the signal propagation at that point. However, if there is an I/O card at the other end of that signal path, the SmartPlant Instrumentation stops the tag signal propagation at the break point and assigns a new connection sequence (100, 99, 98 ...) starting from the I/O card.
- If you replace one signal with another at any point along the wiring path, the software stops the propagation of the old signal at the point where you made the change and propagates a new signal from that point onward to the end of the signal path.

Automatically Propagate a Signal from a Non-Wiring Instrument

This feature makes it possible to associate a conventional or electrical non-wiring instrument with a connected wire, cable set, or cable. Dragging such an instrument from the **Instruments** folder of the **Domain Explorer** to a connected wire, cable set, or cable in the **Connection** window automatically creates a signal from that instrument and propagates this signal along the connected wiring path. Note that non-wiring instruments are instruments that were created based on instrument type profiles that have no wiring.

1. In the **Instruments** folder of the **Domain Explorer**, select a conventional or electrical non-wiring instrument.
TIP Non-wiring instruments are instruments that were created based on instrument type profiles that have no wiring or control system selections but have the **Include wiring** check box selected.
2. Drag the selected instrument to a connected wire, cable set or cable in the **Connection** window.

NOTES

- SmartPlant Instrumentation propagates a signal from the selected instrument along the wiring path based on whether you drag the instrument to a cable, cable set or wire:
 - Dragging an instrument to a cable or a cable set — a signal will be propagated through each of the wires of the cable or the cable set. Each wire will get a different signal level.
 - Dragging an instrument to a single wire — a signal will be propagated only through that wire.

- If the current panel in the **Connection** window is a device panel, the software associates the instrument with the target device panel.
- If the target wires already carry other signals, the software stops these signals and propagates new ones instead.
- The software automatically updates the signal levels based on whether you dragged the instrument to a single wire, a cable, or a cable set. The signal will start at level one and sequence one. However, if level one and sequence one are occupied, the software will automatically look for the next available signal level and sequence.

Create a Local Tag Signal

A local signal is a user-defined signal that does not originate from a device panel or an I/O channel. You create a local signal within any other kind of panel. For example, in an instrumentation electrical interface cabinet, a local signal starts at a terminal relay or in a junction box, it starts at a terminal strip. Note that you can apply tag signals (level and sequence) to wires when no device panel is connected. You can use local signals as follows:

- You can create a local tag signal by assigning a local signal to a wiring tag that does not yet have a signal of its own.
 - You can create a general signal and link it to the signals that are wired into the terminal strip.
1. Do one of the following:
 - In the **Connection** window, click **Actions > Local Signal**.
 - In the **Connection** window, click .
 - On the **Terminal Connection** dialog box, click **Signal**.
 2. On the **Local Signal** dialog box, select a wiring tag that has not been associated with a signal (that is, a tag that has no signal name next to it.)
 3. Click **Create**.
 4. Click **Close** to create and propagate the new local tag signal.

NOTES

- All propagation rules apply to local tag signals.
- For cross wiring with independent wiring from the local terminal strip and I/O card ends, create local tag signals before assigning I/O channels.
- You can apply tag signals (level and sequence) to wires when no device panel is connected.

Filter the Local Signal Dialog Box

This procedure enables you to filter the data window in the **Local Signal** dialog box determining the tag numbers that SmartPlant Instrumentation displays in the **Tag Number** column. This makes it easier to select the tag numbers that you can assign to local signals.

1. On the **Local Signal** dialog box, click **Filter** to open the **Signal Filter** dialog box.
2. Define a conditional expression as follows:
 - a. From the **Column Name** list, select the appropriate tag number attribute according to which you want to filter the data.
 - b. From the **Operator** list, select the required comparison operator to determine how the tag number attribute selected in the **Column Name** field will relate to the expression you enter in the **Value** field.
 - c. From the **Value** data list, select or type the required value to determine how the tag number attribute selected in the **Column Name** field will be specified. You can use wildcards such as % (percent) and _ (underscore) to set the value.

- d. From the **Logical** list, select the required logical operator to determine how the next filter expression will relate to the current one (if applicable).
3. Click **New** to add another data row for an additional filter expression if needed. Make sure you select the appropriate logical operator (**And**, **Or**) at the end of the previous row.
4. To retrieve tag numbers that were imported from SmartPlant Electrical, select **SmartPlant Electrical signals only**.
5. To retrieve tag numbers that were imported from SmartPlant Electrical and that already have an association with a specified power distribution board (PDB) in SmartPlant Electrical, select **Pre-assigned signals only**. This option is available only when you select **SmartPlant Electrical signals only**.
6. To save the filter condition settings so that the next time you open the **Signal** dialog box SmartPlant Instrumentation filters the data accordingly, select **Save filter**.
TIP Click **Restore** to revert to the filter condition that you saved the last time.
7. Click **Verify** to check the correctness of your filter.
8. Click **OK**.

Create a General Signal

A general signal is a user-defined signal which is not characterized by any existing process function and which does not carry any information from an instrument. You link a general signal with tag signals and other general signals. You can use a general signal to treat several tag signals as a group that share common wires. You can use general signals for the following purposes:

- Combine signals into a wire group for transmitting multiple signals that are multiplexed or de-multiplexed through it.
 - Represent the common connection for single or multiple loop power supply.
1. Do one of the following:
 - In the **Connection** window, click **Actions > Local Signal**.
 - In the **Connection** window, click .
 - On the **Terminal Connection** dialog box, click **Signal**.
 2. On the **Local Signal** dialog box, under **General signal**, click **New** in the group box.
 3. On the **New General Signal** dialog box, under **Signal name**, type a unique name.
 4. To associate the new general signal with all the tag signals in the current terminal strip, select **Apply to all tag signals of the current strip**.
 5. Click **OK** to create and propagate the new general signal.

NOTES

- The new general signal name appears in the **Signal** column with no tag next to it. This indicates that this is a general signal that originated in the current terminal strip.
- You can change the general signal name at any time by clicking **Edit** after selecting the required general signal.
- If this general signal is used in a loop drawing, make sure that the general signal name is changed in the loop macro definitions too.

- Macros in a CAD application, that you want to solve with a general signal in SmartPlant Instrumentation, must have the general signal name (as defined in the SmartPlant Instrumentation **Local Signal** dialog box) added as a prefix to the CAD application macro. For example, for the CAD macro **PNL_NAME.1.4**, to be recognized by SmartPlant Instrumentation and solved with the general signal called **GENERALSIG1**, you add the general signal name to the CAD macro as a prefix in the CAD application: **GENERALSIG1.PNL_NAME.1.4**.

Link Tag Signals to a General Signal

This procedure allows you to link tag signals to a general signal that you create on a given strip. You need this functionality to:

- Use a power supply to power tag signal circuits.
 - Multiplex tag signals into a general signal.
1. Recombine tag signals that you previously multiplexed.
 2. Do one of the following:
 - In the **Connection** window, click **Actions > Local Signal**.
 - In the **Connection** window, click .
 - On the **Terminal Connection** dialog box, click **Signal**.
 3. On the **Local Signal** dialog box, under **Signal**, select the general signal to which you want to link tag signals.
 4. Click **Link**.
 5. On the **Link Tag Signals to General Signal** dialog box, under **Link**, select the check boxes for the tag signals that you want to link to the general signal.
 6. Click **Close** to return to the **Local Signal** dialog box.

NOTE Since a general signal does not directly connect to a tag, you need a different way to find the data for the signal's tag numbers. In the drawing block, the macros (tags / attributes) that contain the data for these signals should start with a prefix that reflects the name of the signal.

Multiplex Tag Signals

SmartPlant Instrumentation enables you to multiplex tag signals. That is, you can combine tag signals into a general signal that you propagate. Also, you can combine signals from incoming general signals with independent tag signals into a new general signal that you propagate.

IMPORTANT

- Before you manage the signals within a given terminal strip, create and connect the cables that serve the terminal strip.
 - You can only multiplex tag signals that are wired into the terminal strip.
1. In the **Domain Explorer**, right-click a panel or a terminal strip.
 2. On the shortcut menu, click **Actions > Connection**.
 3. In the **Connection** window, click  next to a terminal in the wire group into which you want to multiplex a signal.
 4. On the **Terminal Connection** dialog box, click **Signal** to open the **Local Signals** dialog box.
 5. Create a unique general signal. For details, see *Create a General Signal* (on page 138).
 6. Link the general signal to the signals that you want to multiplex. For details, see *Link Tag Signals to a General Signal* (on page 139).
 7. Do the following for each terminal in the wire group that you are associating with the general signal:

- a. Under **Signal**, select the general signal that you want to propagate.
 - b. Select a sequence and a signal level.
8. To navigate to other terminals for which you need to change propagation settings, do one of the following:
 - Click **Next** or **Previous**.
 - Under **Terminal number**, select the terminal that you want to edit.
9. Click **Save**.

De-Multiplex Tag Signals

SmartPlant Instrumentation enables you to separate tag signals from a general signal. This is called de-multiplexing. Then, you can propagate the independent signals manually.

IMPORTANT Before you manage the signals within a given terminal strip, create and connect the cables that serve the terminal strip.

1. In the **Domain Explorer**, right-click a panel or a terminal strip.
2. On the shortcut menu, click **Actions > Connection**.
3. In the **Connection** window, click  next to a terminal in the wire group into which you want to de-multiplex a signal.
4. On the **Terminal Connection** dialog box, do the following for each terminal in the wire group that you are associating with one of the de-multiplexed signals:
 - a. Under **Signal**, select the tag signal that you want to propagate.
 - b. Select a sequence as you require. Note that you should increment the sequence according to the original tag signal, rather than according to the general signal with which it was linked.
 - c. Select a signal level.
5. To navigate to other terminals for which you need to change propagation settings, do one of the following:
 - Click **Next** or **Previous**.
 - Under **Terminal number**, select the terminal that you want to edit.
6. Click **Save**.

SECTION 14

Intrinsic Safety

This section describes the procedures for calculating intrinsic safety in SmartPlant Instrumentation.

In This Section

Create an Intrinsically Safe Circuit Type	141
Manage Intrinsic Safety Circuit Types	141
Calculate Intrinsic Safety	142
Define Tag Numbers for Intrinsic Safety Loop Calculation.....	143
Define Circuit Cables for Intrinsic Safety Loop Calculation	143
Calculate Intrinsic Safety for a Loop.....	144

Create an Intrinsically Safe Circuit Type

The first step in making an intrinsically safe calculation is to define a circuit type.

1. Start the Wiring module and do one of the following:
 - Click **Actions > Intrinsic Safety**.
 - Click .
2. On the **Intrinsically Safe Circuit** dialog box, click **New**.
3. Enter the following information:
 - a. Type the required circuit type in the **Circuit Type** data field and the circuit type description in the **Circuit Description** data field.
 - b. From the **I/O Type** list, select the appropriate area I/O type.
 - c. From the **Area Classification** list, select the appropriate area classification.
4. Click **Save**.
5. On the **Drawing Number** dialog box, enter an appropriate drawing number name.
6. Click **OK**.

NOTE The new circuit type is saved and the **Intrinsically Safe Circuit** dialog box opens. You can now select the newly created type from the **Circuit Type** list.

Manage Intrinsic Safety Circuit Types

This option shows you how to maintain the contents of the IS (intrinsic safety) circuit type supporting table from the Instrument Index module. You can also maintain this supporting table from the Wiring module.

1. Open the Instrument Index module by doing one of the following:
 - Click **Module > Instrument Index**.
 - Click .
2. Click **Tables > Intrinsically Safe Circuit Types**.
3. Click **New**, to create a new IS circuit type.
4. Under **Intrinsically Safe Circuit Type**, type the unique and required type name.
5. Under **Intrinsically Safe Circuit Description**, type the description.
6. Under **Drawing Name**, type the drawing name.

7. From the **Area Classification** list, select the area classification.
8. Click **Properties**.
9. On the **Intrinsically Safe Data Input** dialog box, under **Non-hazardous area loop components**, enter the relevant data for the non-hazardous devices, such as barrier, isolator, and so forth. You must define the **R1**, **C1**, and **L1** values.
10. Under **Hazardous area loop components**, enter the relevant data for the hazardous devices, such as transmitters, I/P converters, and so forth. You must define the **R2**, **C2**, and **L2** values.
11. Click **OK** to close the **Intrinsically Safe Data Input** dialog box.
12. Click **OK** to finish.

Calculate Intrinsic Safety

Intrinsic safety calculation in SmartPlant Instrumentation allows you to calculate the maximum permissible cable length between the hazardous and non-hazardous areas based on three main criteria:

- Resistance
- Capacitance
- Inductance

The calculations are made at the domain level.

1. Start the Wiring module and do one of the following:
 - Click **Actions > Intrinsic Safety**.
 - Click .
2. On the **Intrinsically Safe Circuit** dialog box, from the **Circuit type** list, select the appropriate circuit type.

TIPS

- The remaining fields on the dialog box are filled in automatically with the data for the circuit type you select.
 - If the required circuit type is not on the list, create a new one or edit an existing circuit type. For more information, see *Calculate Intrinsic Safety for a Loop* (on page 144).
3. Click **OK**.
 4. On the **Intrinsically Safe Data Input** dialog box, under **Non-hazardous area loop components**, enter the relevant data for the non-hazardous devices, such as barrier, isolator, and so forth. You must define the **R1**, **C1**, and **L1** values.
 5. Under **Hazardous area loop components**, enter the relevant data for the hazardous devices, such as transmitters, I/P converters, and so forth. You must define the **R2**, **C2**, and **L2** values.
 6. Define the cable parameters between the junction box and the marshaling rack.
 7. Define the cable parameters between the device panel and the junction box (if the cable exists). If the field instrument is connected directly to the marshaling rack, specify the cable length (B) = 0.
 8. Click **Calculate** to calculate the limit criteria (resistance, inductance, or capacitance) and the maximum permissible cable length between the junction box and the marshaling rack (Cable A) based on the values you entered.

TIP

- The **Intrinsically Safe Calculation Results** dialog box opens where you can view the calculation results, change the units of measure, and save the results with the different units.

- The maximum permissible length must be less than the actual length of the cable used (Cable “A” – connecting the hazardous and non-hazardous areas).
9. Click **OK** to return to the **Intrinsically Safe Data Input** dialog box.
 10. If desired, you can do any of the following:
 - View and enter revisions.
 - Enter intrinsic safety notes.
 - Click **Print** to generate and print a report that shows all the data that you entered and the calculation results.

Define Tag Numbers for Intrinsic Safety Loop Calculation

When calculating intrinsic safety for a loop, you need to define the tag numbers that belong to the relevant loop. You have to repeat this procedure for each and every tag number in the loop.

1. In the **Domain Explorer**, right- click an instrument, and on the short-cut menu, click **Properties**.
2. On the **Tag Number Properties** dialog box, beside the **Intrinsically safe circuit type** list, click .
3. On the **Intrinsically Safe Circuit Types** dialog box, select the required row, and then click **Properties**.

TIP You can also create a new intrinsically safe circuit type.
4. On the **Intrinsically Safe Data Input** dialog box, in both group boxes, make sure that you express the following parameters in the following units of measure:
 - **Resistance** — ohm
 - **Inductance** — mH
 - **Capacitance** — μF (microFarad)
 - **L/R** — mH/ohm
5. Click **OK** to close the **Intrinsically Safe Data Input** dialog box.
6. Make sure that the row that you require is selected and then click **OK** to close the **Intrinsically Safe Circuit Types** dialog box.
7. On the **Tag Number Properties** dialog box, click **OK**.

Define Circuit Cables for Intrinsic Safety Loop Calculation

Before calculating intrinsic safety for a loop, you must perform this procedure for each cable in the circuit.

1. In the **Domain Explorer**, right- click a cable, and on the short-cut menu, click **Properties**.
2. On the **Cable Properties** dialog box, beside the **Type** list, click .
3. On the **Cable Types** dialog box, select the required row, and then click **Properties**.

TIP You can also create a new intrinsically safe circuit type.
4. On the **Cable Type Properties** dialog box, make sure that you express the following parameters in the following units of measure:
 - **Capacitance** — nF (nanoFarad) / km
 - **Resistance** — Ohm/km
 - **Inductance** — mH/km
5. Click **OK** to close the **Cable Type Properties** dialog box.

6. Make sure that the row that you require is selected and then click **OK** to close the **Cable Types** dialog box.
7. On the **Cable Properties** dialog box, under **Unit of measure**, select **meter**.
8. Under **Length**, define the cable length.
9. Select the **Set as intrinsically safe** check box, and then click **OK**.

Calculate Intrinsic Safety for a Loop

After defining loop tags and circuit cables as required, use this procedure to perform the loop intrinsic safety calculation.

1. Start the Wiring module and click **Actions > Intrinsic Safety Loop Calculation**.
2. On the **Enter Loop Number** dialog box, do one of the following:
 - Type the loop number in the data field.
 - Click **Find** to search for a loop that requires intrinsic safety calculation.
3. Click **OK** to open the **Intrinsically Safe Loop Data** dialog box.

TIPS

- In the data window, if you select a tag for which you did not define an intrinsically safe circuit type, nothing is displayed in the lower group box.
 - If you select a tag number for which you defined an intrinsically safe circuit type, SmartPlant Instrumentation displays non-hazardous area and hazardous area loop components below the data window.
4. On the **Intrinsically Safe Loop Data** dialog box, click **Calculate**.
 5. In the **Intrinsically Safe Calculation Results** dialog box, select each tag number for which you want to display intrinsic safety data.
 6. To preview reports for all of the tags in the current loop, click **Print**.

SECTION 15

Cable Routing

The cable routing feature provides the ability to set up a flexible, modular model of the cable routing in your <plant>. Cable routing sections are divided into several categories that enable you to set up a complete cable routing sequence. These categories match the different <plant> regions as follows:

Category	Category Description and Location in the <Plant>
Trunk	Standard sections that include all the cable routing that does not pass through a built area. These sections are actually legs passing through the <plant> junction points that function as ports, where cables enter and exit the routing. Trunks contain trays conveying the positions through which the cables pass.
Building	Sections that pass through a built area. Building sections do not contain trays or positions.
Panel – Routing Distance	This in fact, is not a section category but a variable distance added to the total cable routing length. It is the distance between an instrument and the routing starting point.

In an Owner Operator domain (As-Built), it is possible to claim cables and their cable routing data to a project. Once claimed to a project, the routing data is no longer available to work within As-Built, until the data is merged back. You can only work on cable routing in one project at a time. Reports and validations are carried out on the plant and projects as if they were one. To work with cable routing in another project, you must first merge the project with the routing data back to the plant, this removes the routing data from the project and replaces the data in the As-Built. Then reclaim making sure that the first project you claim to is the project that you want to work with cable routing.

In an Owner Operator plant you can create Cable Routing Reports that contain all the cable routing data from the As-Built and all projects.

Cable Routing Flow of Activities

The cable routing feature allows you to create modular cable routing sections. The sections convey the cables in your <plant>, and they subdivide into two main categories:

- A **Trunk** is a standard section that can be utilized in all non-enclosed plant areas. Trunks contain positions, fixed on trays.
- A **Building** is a section that passes through an enclosed area, and does not contain positions

You plan your cable routing system according to the following flow of activities:

1. Set the cable routing options, such as cable length and position width units of measure, cable spare length, and so forth. For details, see *Set Cable Routing Options* (on page 146).
2. Create named standard widths. You will use these later to create positions in the trunk section. For details, see *Manage Standard Widths* (on page 147).
3. Create standard positions. Positions convey the cables within the trunk sections. For details, see *Manage Standard Positions* (on page 147).

4. Define the maximum number of cables for each standard position, according to the position width. This is how you assure that the number of cables does not exceed the position cable capacity. For details, see *Set the Maximum Number of Cables for the Standard Positions* (on page 148).
5. Create the routing sections. For details, see *Create a Routing Section* (on page 148).
6. Associate cables with sections. The association can be performed in single or batch mode. The association includes the following activities:
 - Selecting the cables to which you add the cable routing.
 - Setting the order of the routing sections per cable.
 - Selecting the active positions for each section.

Set Cable Routing Options

Use this procedure to set the options for the cable routing throughout a specific <plant>. SmartPlant Instrumentation applies your selections to existing cables and sections and to all future routing elements. When you change the **Length unit of measure**, the software automatically recalculates values of routing elements that currently exist in your plant and expresses them in the new units of measure. The routing elements that SmartPlant Instrumentation recalculates after a change in the **Cable Routing Options** dialog box are cable length, panel-routing distance, and section length.

1. In the **Wiring Module** window, click **Actions > Cable Routing Options**.
2. On the **Cable Routing Options** dialog box, from the **Length unit of measure** list, select a unit of measure.
3. In the message box that opens, click **OK** to confirm calculation in the new unit of measure.
4. From the **Position width unit of measure** list, select the required unit of measure.

TIP Set the **Position width unit of measure** at initialization, and do not change it. Although changing the length unit of measure recalculates length values in the new unit of measure, changing the position width unit of measure changes the label without recalculating the value of the position width.
5. In the message box that opens, click **OK** to confirm the position width unit of measure.
6. In the **Cable spare length** group box, to calculate the spare cable length for all the cables in the <plant>, do one of the following:
 - Select **Fixed length** and type its value in the box to the right.
 - Select **Fixed percentage**, and type or select a percent value in the box to the right.
7. Under **Panel to routing distance**, type the distance from a junction box or marshaling rack to the beginning of a routing section.
8. To customize cable routing terminology, next to **Customize terminology**, select the check box and then click the **Terminology** command button. For details, see *Customize Cable Routing Terminology* (on page 147).

NOTE For cable drums with status **Design** or **Purchased**, if you change cable spare length, SmartPlant Instrumentation recalculates the cable length, but not for drums with status **Locked**.

Customize Cable Routing Terminology

Use this procedure to change cable routing terminology to terms that you define. You can change any or all of the following terms: trunk section, building section, position, section, and tray. Changes that you make are implemented in all relevant dialog boxes and reports.

1. In the **Wiring Module** window, click **Actions > Cable Routing Options**.
2. On the **Cable Routing Options** dialog box, select **Customize terminology** and then click **Terminology**.
3. On the **Cable Routing Custom Terminology** dialog box, for each term that you want to customize, under **Custom Heading (Singular)** and **Custom Heading (Plural)**, type the desired values.
4. Click **OK** to close the **Cable Routing Custom Terminology** dialog box.
5. Click **OK** to close the **Cable Routing Options** dialog box.

Manage Standard Widths

The standard width is a value that you enter and which SmartPlant Instrumentation uses to define the width of the routing positions that the sections comprise. This procedure explains how to add a standard width to your current <plant>, edit, or delete an existing one. Note that a standard width is not necessarily associated with a position. It only provides the ability to associate the width with positions.

1. In the **Wiring Module** window, click **Tables > Cable Routing > Standard Widths**.
2. On the **Standard Widths** dialog box, do one of the following:
 - To add a new standard width, click **New**.
 - To edit an existing standard width, click the field that you want to edit.
 - To delete a standard width, highlight the required standard width or in the **Find** field, type the width that you want to delete, and click **Delete**.
3. Type the required standard width value.
4. Click **OK** to save your changes and close the dialog box.

Manage Standard Positions

Routing positions are the wiring elements that convey the cables in your <plant> and separate the different cable groups inside the routing sections. You associate the standard routing positions with the trunk sections in your <plant> and adjust their widths to fit those positions. This way you do not have to define new positions for every new trunk that you add.

1. In the **Wiring Module** window, click **Tables > Cable Routing > Standard Routing Positions**.
2. On the **Standard Routing Positions** dialog box, do one of the following:
 - To add a new standard position, click **New**.
 - To edit an existing standard position, click the field that you want to edit.
 - To delete a standard position, highlight the required standard position or in the **Find** field, type the **position** that you want to delete, and click **Delete**.
3. Type the position name and description as required..
4. Repeat steps 1-3 for all the standard positions you want to enter.
5. Click **OK** to save your changes and close the dialog box.

Set the Maximum Number of Cables for the Standard Positions

The maximum number of cables is a value that you define to limit the number of cables that you can associate with a position within a trunk section. You can add positions to trunks only if you predefined a maximum number of cables per position. That is, you define the number of cables for a standard position and only then the position becomes available to include in a trunk.

1. In the **Wiring Module** window, click **Tables > Cable Routing > Maximum Number of Cables**.
2. On the **Maximum Number of Cables** dialog box, click a field and type the maximum number of cables in the width column of the required position.
TIP The **Maximum Number of Cables** dialog box comprises a grid. The grid displays the standard positions and standard widths that you have added to your current <plant>.
3. Click **OK** to save your changes and close the dialog box.

Create a Routing Section

Routing sections are the wiring elements that convey the cables in your <plant>. They subdivide into two main categories:

- **Trunk** — a standard section that can be utilized in all non-enclosed <plant> areas. Trunks contain positions.
- **Building** — a section that passes in an enclosed area and does not contain positions.

You define a standard position with a standard width to fit the section that includes it.

1. In the **Wiring Module** window, click **Tables > Cable Routing > Routing Sections**.
2. On the **Routing Sections** dialog box, do one of the following:
 - To base the new routing section on an existing routing section, click **Duplicate**.
 - To create a new routing section not based on an existing section, click **New**.
3. In the **Routing Section Properties** dialog box, type the new section name or accept the default in the **Routing section** field.

TIP The initial **All** or **Trunk** default is **T-1**, and the initial **Building** default is **B-1**. Defaults for succeeding entries revise the previous name in a given category by adding a 1 or incrementing the previous suffix by 1.

4. In the **Length** field, type the section length. To change the unit of measure, see *Set Cable Routing Options* (on page 146).
5. From the **Routing category** list, select **Trunk** or **Building**.
6. If you selected **Trunk**, SmartPlant Instrumentation displays the **Position** and widths grid. Select a defined width for each active position.

TIP When you define a maximum number of cables for a standard position, the position at the specific defined width becomes available for association with sections. In the **Position-Width** grid of the **New Section** dialog box, the available positions and widths are marked with a check box. Although a position can have more than one available standard width, you can select only one standard width per position.

7. Click **OK** to enter the new section and close the dialog box.

Edit a Routing Section

This procedure explains how to edit the properties of a routing section.

1. In the **Wiring Module** window, click **Tables> Cable Routing > Routing Sections**.
2. On the **Routing Sections** dialog box, filter the routing sections by selecting **Trunk** or **Building**, or accept the default **All**.
3. Highlight a routing section and click **Properties**.
4. On the **Routing Section Properties** dialog box, you have the following options:
 - To modify the routing section name.
 - To modify the length (unless the cable assigned to this section is on a locked drum).
 - To change the section category, from the **Routing category** list, if the section is not in use.
 - For routing category **Trunk**, to select a width for each position from the **Position and widths** grid (unless the cable assigned to this section is on a locked drum).
5. Click **OK** to save your changes and close the **Routing Section Properties** dialog box.

Associate a Routing Section with Cables

The last stage in setting up cable routing is associating the different sections that you have created with the cables in your <plant>. Once the association is completed, SmartPlant Instrumentation calculates and sets the total length of the routing, and the cable routing of your <plant> is ready.

1. In the **Domain Explorer**, highlight the cables with which you want to associate cable routing sections.
2. Right-click the highlighted cables, and then on the shortcut menu, click **Actions > Cable Routing**.
3. On the **Cable Routing** dialog box, from the **Cable** list, select a cable.
4. In the **Available Sections** window, highlight the available sections that you want to associate with the selected cable, and do one of the following:
 - Click 
 - Drag the sections to the **Assigned sections** pane.
5. From the **Routing Position** list of the required section, select the position that will hold the cable.

TIP Change the order of the assigned sections by highlighting an assigned section and clicking **Move Up** or **Move Down**.
6. Type the panel-routing distance, or accept the default.
7. Repeat steps 2 through 5 for each cable in the **Cable** list.

Edit a Cable Routing

This feature allows you to change the sections associated with a cable, change their order, or change the panel-routing distance.

1. In the **Domain Explorer**, highlight the cables whose routing you want to edit.
2. Right-click the highlighted cables, and then on the shortcut menu, click **Actions > Cable Routing**.
3. On the **Cable Routing** dialog box, from the **Cable** list, select a cable.
4. To add additional sections to the selected cable, under **Available sections**, highlight the available sections that you want to associate with the cable, and do one of the following:
 - Click .
 - Drag the sections to the **Assigned sections** pane.
5. To remove sections from the selected cable, under **Assigned sections**, highlight the sections that you want to remove, and do one of the following:
 - Click .
 - Drag the sections from the **Assigned sections** pane to the **Available sections** pane.
 - Click **Un-assign**, to remove all the sections from the **Assigned sections** pane.
6. Change the position of a section, as necessary, from the **Routing Position** list.
7. Change the order of assigned sections, as necessary, by highlighting a section and clicking **Move Up** or **Move Down**.
8. Edit the panel-routing distance, as necessary, by changing the value in the **Panel- routing distance** box.
9. Repeating steps 3 through 7 for each cable in the **Cable Name** list.

Copy Routing Data to Another Cable

After you create a cable routing, you can copy the routing data to other cables, thus saving redefinition time for similar cables. Note that if you copy routing data to a cable with existing routing, the new routing overwrites the existing routing data.

1. In the **Domain Explorer**, select the cables from which you want to copy routing data.
2. Right-click the highlighted cables, and then on the shortcut menu, click **Actions > Copy Cable Routing**.
3. On the **Copy Cable Routing** dialog box, select a cable from the **Wiring Explorer** list.
TIP A source cable is a trunk cable for which you have already defined cable routing options, created the routing sections, and associated cables with these sections. SmartPlant Instrumentation displays the routing data for the source cable you select in the **Source routing data** grid.
4. From the **Wiring Explorer**, highlight target cables, and drag them to the **Destination cables** group box.
5. Select the cables to which you want to copy the routing data of the source cable in one of the following ways:
 - Select the check box to the left of each cable.
 - Choose **Select all cables without routing**. Use this option to avoid overwriting existing routing data.
 - Choose **Select all** to enable overwriting of existing routing data on the destination cables.
6. Click **Copy**.

NOTE To copy routing data to additional cables, repeat steps 3 through 6.

Append Routing Data to a Cable

You can append the routing properties of an existing source cable to a destination cable or cables. This can be an efficient way of adding sections to an existing routing.

- A source cable is a trunk cable for which you have already defined cable routing options, created the routing sections, and associated cables with these sections.
 - A destination cable is a trunk cable to which you want to add the routing data of the source cable.
1. In the **Domain Explorer**, select the cables that will be your source for appending.
 2. Right-click the highlighted cables, and then on the shortcut menu, click **Actions > Append Cable Routing**.
 3. On the **Append Cable Routing** dialog box, select a cable from the **Wiring Explorer** list.
TIP In the **Source routing data** group box, SmartPlant Instrumentation displays the routing data for the selected cable.
 4. In **Wiring Explorer**, highlight target cables, and drag them to the **Destination cables** group box.
 5. Select the cables to which you want to append the routing data of the source cable in one of the following ways:
 - Select the check box to the left of each cable.
 - Choose **Select all cables without routing**. Use this option to avoid overwriting existing routing data.
 - Choose **Select all** to enable overwriting of existing routing data on the destination cables.
 6. Click **Append**.

NOTES

- To append routing data to additional cables, repeat steps 3 through 6.
- SmartPlant Instrumentation appends the new sections after the original sections of the destination cable, if there were any. You can change the order of these sections by opening the destination cable in the **Cable Routing** dialog box. For details, see *Edit a Cable Routing* (on page 150).

SECTION 16

Cable Drums

IMPORTANT It is recommended that any cables in the As-Built, that have been assigned to cable drums, are un-assigned from their drums before you claim them to the project. Failure to do so, will result in inconsistencies in the As-Built drum data when merging the cables back to the As-Built.

The cable drum feature allows you to efficiently use cable drums, and setup an organized method of cable drum assignment. The cable drum feature handles both the optimization of existing cable drum allocations (purchased drums) and the assignment of new cable drums when necessary (Design drums). The cable drum feature allows you to perform cable drum assignment both automatically and manually. The cable drum feature requires that you define pulling areas in your <plant> where the cable drums are concentrated. This can be done in an Engineering plant, Owner Operator plant (As-Built), or project. From these areas you perform the assignment of cable drums for the cables.

The pulling area that you define here does not have the same functionality as the <area> in SmartPlant Instrumentation plant hierarchy. The pulling area is designated for cables and cable drums only.

The steps required to set up a fully functional cable drum assignment are as follows:

1. Define the pulling areas where the cable drums are placed.
2. Associate cables with the predefined pulling areas.
3. Define drum attributes for cable types.
4. Create the cable drums that will eventually be assigned to your cables. This includes both adding the cable drums that already exist in your <plant> and adding the cable drums that you need to purchase to complete your required cable drum inventory. You can also perform this automatically in a cable drum assignment.
5. Select the cables that you want to assign to the cable drums that you have created (even if the drums have not been purchased yet).
6. The last stage is to assign cables to cable drums. At this stage the cable drum feature automatically adds new cable drums as required.

NOTES

- It is recommended that when working in an As-Built plant with projects, you carry out the workflow for creating cable drums from within the project.
- Pulling areas are common to the As-Built and all its projects.

Create and Manage a Pulling Area

The pulling area is the plant area where you concentrate cable drums. This procedure explains how to add a new pulling area to your <plant>. Use this procedure to edit existing pulling area data.

1. In the **Wiring Module** window, click **Tables > Pulling Areas**.
2. On the **Pulling Areas** dialog box, do one of the following:
 - To create a new pulling area, click **New**.
 - To edit a pulling area, click the field that you want to edit.
 - To delete a pulling area, select a row and click **Delete**.
3. Type the pulling area name (required) and description in the respective fields.
4. Click **OK**.

NOTE The pulling area that you define here does not have the same functionality as the <area> in SmartPlant Instrumentation plant hierarchy. The pulling area is designated for cables and cable drums only.

Associate Cables with a Pulling Area

Prior to assigning cables to drums, you have to associate your cables with a pulling area.

1. In the **Domain Explorer** window, select and right click the cable that you want to associate with a pulling area.
2. On the shortcut menu, click **Properties**.
3. On the **Cable Properties** dialog box, from the **Pulling Areas** list, select the pulling area that you want the selected cable to be associated with.

TIP You can use the **Pulling Areas** dialog box to add new pulling areas or edit existing.

4. Click **OK** to save your changes and close the **Cable Properties** dialog box.

Define Cable Drum Attributes for Cable Types

This procedure explains how to define cable drum attributes for cable types. This is required because each cable type has its minimum and maximum length. The cable drums that you will create will then be based on the cable drum attributes that you define here.

1. In the **Wiring Module** window, click **Tables > Cable > Types**.
2. On the **Cable Types** dialog box, select a cable type and click **Properties**.
3. On the **Cable Type Properties** dialog box, type the appropriate data in the following fields:
 - **Maximum drum length** — type the maximum cable drum length for this cable type.
 - **Minimum drum length** — type the minimum cable drum length for this cable type.
 - **Length UOM** — select the length unit of measure.
 - **Drum spare length percent** — enter the percent of the used cable drum length to allocate to spare cable length. (The spare and the used length combine to the total length.)
4. Click **OK** to save your changes and close the **Cable Type Properties** dialog box.
5. Click **OK** to close the **Cable Types** dialog box.

Manage Cable Drums Manually

This procedure explains how to create cable drums manually and set their attributes. There are three categories of cable drums:

- **Design** — Cable drums that you intend to purchase in the future and you want to design to fit your cables.
 - **Purchased** — Existing cable drums that you want to include in a cable-drum optimization.
 - **Locked** — Existing cable drums that you do not want to include in a cable-drum optimization.
1. In the **Wiring Module** window, click **Tables > Cable Drums**.
 2. On the **Cable Drums** dialog box, select a cable type to filter the data window.
 3. Do one of the following:
 - Click **New** to add a new cable drum.
 - Select a cable drum row and click in the appropriate fields to edit the data.
 - Select a cable drum row and click **Delete**.

4. In the data window, type the data in the fields or select the data from the lists according to the following table:

Column Name	Column Description
Name	Type the cable drum name.
Description	Type the cable drum description if required.
Pulling Area	Select the pulling area where the cable drum is located.
Length	Type the total cable drum length (utilized + spare).
Utilized Length	Displays the utilized cable length of the total drum length.
Length UOM	Select the unit of measure used to measure the drum length.
Drum Status	Select one of the following: Design — cable drum in the design stage (not purchased yet); available for cable-drum optimizations. Purchased — a purchased cable drum of defined length that you want to include in cable-drum optimizations. Locked — a purchased cable drum that is assigned to a cable and locked against reassignment (cable drum optimizations).
Spare Length Percent	Type the percent of the used cable drum length to allocate to spare cable length.

5. Click **OK** to save your changes and close the **Cable Drums** dialog box.

NOTE SmartPlant Instrumentation creates cable drums automatically when you assign cables to cable drums and additional cable drums are required.

Assign Cables to Drums Automatically (with Optimization)

When assigning cables to cable drums, SmartPlant Instrumentation creates cable drums automatically. You can perform this assignment and optimization in single or in batch mode. The optimization process matches cables and the cable drums that fit them with minimum waste of cable drum length, using the following algorithm:

- The highest priority is to utilize cable drums that are purchased but not yet locked. Optimization may assign new cables to these drums.
- The second priority is to optimize the drum cable length for drums whose status is design. The guiding parameters are minimum drum length and maximum drum length.
- The third priority is to create new cable drums of a given cable type.

IMPORTANT The procedure below only works if you created a cable and assigned a pulling area.

1. In the **Wiring Module** window, click **Actions > Cable-Drum Assignment**.
2. On the **Cable-Drum Assignment** dialog box, select the following filtering criteria:
 - Cable type
 - Pulling area
3. Do one of the following:
 - Select **Include assigned cables** to display cables that are already assigned to cable drums, in addition to those not yet assigned.

- Clear **Include assigned cables** not to display assigned cables.
- 4. Click **Find**.
- 5. Select the cables that you want to include in the cable drum assignment.
TIP If you selected **Include assigned cables** in step 3 above, optimizing can change existing cable assignments.
- 6. Click **Optimize**.

Assign Cables to Drums Manually (Without Optimization)

This procedure explains how to assign a cable to a cable drum manually. This action requires that you predefine the following for this cable:

- A cable type.
- At least one pulling area.
- A cable drum associated with the cable type and with the pulling area.

The manual cable drum assignment is performed without optimization, which means that the designated cable drum is not necessarily the optimum for the selected cable.

1. In the **domain Explorer**, right-click a cable and then click **Properties**.
2. On the **Cable Properties** dialog box, do the following:
 - a. From the **Pulling area** list, select the pulling area that the designated cable drum is associated with.
 - b. Make sure that the cable type is selected from the **Type** list.
 - c. From the **Cable Drum** list, select the cable drum to which you want to assign the selected cable. Click  to add any required items that are not on the list.
 - d. Click **OK** to save your changes and close the **Cable Properties** dialog box.

SECTION 17

Managing Panel Locations

Panel location is a panel property that you define when creating or editing the properties of a panel. You can set multiple location levels, for example, building — floor — room. You can use each level to define a panel location.

In This Section

Define a New Panel Location	157
Modify Panel Location Properties.....	157
Change the Location of a Panel	158
Dissociate a Panel from a Location	158
Delete a Panel Location.....	158

Define a New Panel Location

This option allows you to define a new panel location. You can then select this location when defining panel properties. Since panel location is defined per plant, all the panel location definitions that have been made in the Administration module are available throughout the plant.

1. Do one of the following to open the **Location Manager** dialog box:
 - In the Wiring module, click **View > Location Manager**.
 - Click  next to the **Location** list arrow in any appropriate **Properties** dialog box.
2. Do one of the following in the **Location Manager**:
 - To define a location on the highest hierarchy level, right-click **Location**, and then on the shortcut menu, click **New**.
 - To define a location under an existing location definition, expand one of the existing location levels, then right-click a location and on the shortcut menu, click **New**.
3. On the **Location Properties** dialog box, type the location name and description as desired.

NOTE You can define a new location on the highest hierarchy level directly from the **Domain Explorer**. Right-click the **Panels by Location** folder, and then on the shortcut menu, click **New > Location**.

Modify Panel Location Properties

This option allows you to modify the properties of a panel location. You can rename a location and change its description.

1. Do one of the following to open the **Location Manager** dialog box:
 - In the Wiring module, click **View > Location Manager**.
 - Click  next to the **Location** list arrow in any appropriate **Properties** dialog box.
2. Right-click a location and then on the shortcut menu, click **Properties**.
3. On the **Location Properties** dialog box, type the location name and description as you require.
4. Click **OK**.

Change the Location of a Panel

You can change the location of a panel either in the **Domain Explorer** by dragging a panel to another location in the **Panels by Location** folder or in the **Location Manager**.

1. On the **Properties** dialog box of a panel, click  next to the **Location** list arrow.
2. In the **Location Manager**, select a location and click **OK**.

TIP To dissociate a panel from a location without assigning another location, click **Dissociate**.

3. Click **OK** on the **Panel Properties** dialog box.

NOTE Also, you can change the location of a panel in the **Domain Explorer** without opening the **Properties** dialog box for a panel. Expand the **Panels by Location** folder and select a panel. Drag it to another location, as you require.

Dissociate a Panel from a Location

While defining panel properties, you can dissociate a panel from a location.

1. In the **Domain Explorer**, right-click a panel.
2. On the shortcut menu, click **Properties**.
3. Click  next to the **Location** list arrow.
4. In the **Location Manager**, click **Dissociate**.

Delete a Panel Location

This procedure explains how to delete a panel location in the **Location Manager**.

1. Do one of the following to open the **Location Manager** dialog box:
 - In the Wiring module, click **View > Location Manager**.
 - Click  next to the **Location** list arrow in any appropriate **Properties** dialog box.
2. Right-click a location.
3. On the shortcut menu, click **Delete**.

NOTE You can delete a location directly from the **Domain Explorer** without opening the **Location Manager**. In the **Domain Explorer**, expand the **Panels by Location** folder. Then, right-click a location and then on the shortcut menu, click **Delete**.

SECTION 18

Cable Block Diagrams

Cable Block Diagrams (CBDs) provide a graphical representation of the interconnection of the cables and the various junction boxes, marshaling racks, cabinets, wiring equipment, and so forth in your project. They show the project cable flow between the various cables, cabinets, and wiring equipment of your project.

Cable Block Diagrams are ideal for use in the preliminary planning stages of your plant to verify the destination panels for cables as no wiring connections or terminations are needed at this early stage of your project. A CBD is also used for ordering made to order (MTO) cables at the feed stage of your project, and later after completion of your project, as part of the maintenance program to show the location of all major cables, panels, junction boxes, and so forth.

SmartPlant Instrumentation allows you to drag and drop panels and wiring equipment from the Domain Explorer to your drawing and position them exactly where you want. The software creates a representation of existing cables on the fly, or you can manually connect cables between item connection points.

NOTE When dragging 2 connected panels or wiring equipment to a CBD, the connecting cable is not displayed in the CBD folder under the CBD drawing, until you save the CBD using the **File > Save Custom Changes** command.

Create a Cable Block Diagram

This procedure shows you how to create a new cable block diagram, add items, show already connected cables, and connect cables between the items. Using symbols supplied with the software, or symbols created in the Symbol Editor, you drag and drop items where you want to position them within the drawing. When you connect cables between the different items of equipment, the software checks that the connection point is valid, and makes the connection. If no connection point is available the software does not allow the cable to be connected.

IMPORTANT You must associate panels and other items with a symbol to display them on the CBD,. For more information, see *Associate a Symbol with an Item* (on page 23).

1. On the **Domain Explorer**, locate the **Drawings** folder.
2. In the Drawings folder, locate the **Cable Block Diagram** folder.
3. Right-click the **Cable Block Diagram** folder, and from the shortcut menu select **New > Block Diagram**.
4. On the **Document Properties (New)** dialog box, type a value for the **Document number** and **Description** fields.
5. Click **OK**.
6. In the **Cable Block Diagram** folder, right-click your new document, and from the shortcut menu select **Reports > Generate Cable Diagram**.
7. At the prompt, click **Yes** to preview your drawing, a new blank drawing opens in the Enhanced Report Utility.
8. In the Enhanced Report Utility, open the **Domain Explorer** by doing one of the following:
 - Press F7
 - Click **Tools > Domain Explorer**
 - Click 
9. From the **Domain Explorer** in the Enhanced Report Utility drag the panels you require, and position them on the drawing.

TIP The order that you drag the panels onto the drawing will define the sorting order that will be used in the generation process of the connection, for example; if you drag the DCS panel first, then the JB, and finally the device panel, the software will connect the DCS to the JB and then the JB to the device panel.

10. Click  to allow the software to automatically regenerate your drawing, and show any cables and their connections between the physically wired items.

NOTE This applies to panels and cables that have already been associated together. For more information, see *Associate a Cable with a Panel* (on page 161).
11. To make the connections between the items manually, do the following:
12. Select any item on the drawing and do one of the following:
 - Click **Activate Connection Mode** .
 - Click **Edit > Activate Connection Mode**.
13. On the **Domain Explorer** select the cable you want to connect between two items.
14. Move the cursor over the drawing to view the available, valid connect points of the items and cable category on your drawing.
15. Click the connect point of the item you want the cable to connect 'From'.
16. Click the connect point of the item you want the cable to connect 'To'.
17. Select another cable or click **View > Refresh** to view the drawing with the cable attached.

Add Cables and Objects to an Existing Cable Block Diagram

This procedure shows you how to add cables and items to an existing cable block diagram.

IMPORTANT You must associate panels and other items with a symbol to display them on the CBD,. For more information, see *Associate a Symbol with an Item* (on page 23).

1. Open the CBD that you want to add items to. For further details, see *View an Existing Cable Block Diagram* (on page 161).
2. In the Enhanced Report Utility, open the **Domain Explorer** by doing one of the following:
 - Press F7
 - Click **Tools > Domain Explorer**
 - Click 
3. From the **Domain Explorer** in the Enhanced Report Utility drag the objects you want to add to the drawing, and position them on the drawing.
4. Click  to allow the software to automatically regenerate your drawing, and show any cables and their connections between the physically wired items.

NOTE This applies to panels and cables that have already been associated together. For more information, see *Associate a Cable with a Panel* (on page 161).
5. To make the connections between the items manually, do the following:
 - a. Select any item on the drawing and do one of the following:
 - Click **Activate Connection Mode** .
 - Click **Edit > Activate Connection Mode**.
 - b. On the **Domain Explorer** select the cable you want to connect between two items.
 - c. Move the cursor over the drawing to view the available, valid connect points of the items and cable category on your drawing.

- d. Click the connect point of the item you want the cable to connect 'From'.
- e. Click the connect point of the item you want the cable to connect 'To'.
6. Select another cable or click **View > Refresh** to view the drawing with the cable attached.

View an Existing Cable Block Diagram

This procedure shows you how to view an existing cable block drawing.

1. On the **Domain Explorer**, locate the **Drawings** folder.
2. In the Drawings folder, locate the **Cable Block Diagram** folder.
3. In the **Cable Block Diagram** folder, right-click the drawing you want to view, and from the shortcut menu select **Reports > Generate Cable Diagram**.
4. At the prompt, click **Yes** to preview your drawing, the drawing opens in the Enhanced Report Utility.

Disconnect Cables from Items in a Cable Block Diagram

This procedure shows you how to disconnect cables from items on your drawing.

1. Open the CBD that you want to disconnect cables from. For further details, see *View an Existing Cable Block Diagram* (on page 161).
2. Select the cable you want to disconnect.
3. Do one of the following:
 - Click .
 - Click **Edit > Disconnect**.
 - Right-click, and on the shortcut menu, click **Disconnect Items**.
4. At the prompt, select the required check box, and click **OK**.

NOTES

- If the cable for disconnection is only connected to one item, no prompt will appear and the cable is disconnected automatically.
- Disconnecting both sides of the cable results in the loss of any wiring associated with the connection.

Associate a Cable with a Panel

You use the **Associate Cable with Panels** feature in conjunction with cable block diagrams, to associate a specific cable to specific panels. When you use the cable in your cable block diagram the panels that are associated with the cable are automatically placed on your drawing along with the cable.

By dragging panels from the Domain Explorer to the **Associate Cable with Panels** window you can associate the panel with either side of the cable. You can see which panels are already associated or connected to the cable and to what side of the cable the panel is associated. Also you can move a panel from one side of the cable to the other.

1. In the **Domain Explorer**, navigate to the **Cables** folder.
2. Open the **Cables** folder, and select the cable you want to associate with a panel.

TIP You can choose more than one cable at the same time by clicking the required cables in the **Items** pane in the Domain Explorer. In the **Associate Cable with Panels** window, you choose the cable that you want to associate to a panel from the **Cable** pick list in the window.

3. Right-click the cable, and from the shortcut menu, click **Actions > Associate Panels**, the **Associate Cable with Panels** window opens.

TIP The **Associate Cable with Panels** window shows the physically connected panels as well as the associated panels. A cable that is already associated to a panel, displays



next to the panels name. A cable that is physically connected to a panel, displays



next to the panels name.

4. In the **Domain Explorer**, navigate to the **Panels by Category** folder.
5. Locate the required panel you want to associate with the cable.
6. From **Panels by Category** folder, drag the required panel icon on to the **Cable End One** or **Cable End Two** column of the **Associate Cable with Panels** window.

TIPS

- You can drag the same panel into the window a number of times, providing that it is to the same **Cable End** each time.
 - You can also select an item in the **Associate Cables with Panels** window, right/click, and from the shortcut menu use the **Copy/Paste** commands to copy and paste an item into a new row.
 - To change a panels cable end association, drag the panel from one cable end column to the other cable end column.
7. Close the **Associate Cable with Panels** window, click **Yes** at the prompt to save the associations.

NOTES

- A cable must have a panel associated at each end to save the associations.
- Cable relationships which were manually associated with panels are not removed or deleted by the software when physically connected with the panels.
- The relationship between the physically connected panels and the cable shown in the **Associate Cable with Panels** window cannot be deleted. They are removed when the cable is disconnected.
- The **Associate Cable with Panels** window does not display, or take into consideration, pre-assigned panels or junction boxes that were assigned using instrument connection pre-assignment browsers.

Creating Wiring Equipment for Cable Block Diagrams

To create wiring equipment for a cable block diagram, you must carry out the following procedures:

1. Create a symbol containing the connectors, macros, and so forth for the required wiring equipment. For more information, see *Creating a Wiring Equipment Symbol Assembly* in the *Enhanced Report Utility help* or *Symbol Editor help*.
2. Create the wiring equipment in the **Domain Explorer** and associate the wiring equipment assembly symbol with the wiring equipment. For more information, see:
 - *Create a Wiring Equipment Item* (on page 64).
 - *Associate a Symbol with an Item* (on page 23).
3. Right-click the wiring equipment, and on the shortcut menu, click **New > Plug-and-Socket Group**.
4. Follow the instructions on the **Plug-and-Socket Group** wizard, to create the connector groups. For more information, see *Plug and Socket Group Wizard* (see "*The Plug-and-Socket Group Wizard*" on page 77).

NOTE At the end of the process, under the wiring equipment, the **Plug-and-Socket Group** icon  appears.

5. Connect the required cables to the wiring equipment. For more information, see *Connect a Cable to a Plug and Socket Group* (on page 78).

NOTE To connect a cable to the plug-and-socket group, the cable must have a plug-and-socket connector on one end of the cable.

See also

Create a Cable Block Diagram (on page 159)

Filtering Items on a Cable Block Diagram

You can filter items on a Cable Block Diagram by creating a filter in the **Filter Definition** dialog box for the specific CBD.

1. In the **Domain Explorer**, open the **Cable Block Diagrams** folder located in the **Drawings** folder.
2. Right-click on the CBD that you want to apply a filter to, and from the shortcut menu select **Filter**.
3. On the **Filter Definition** dialog box, under **Filter name**, type a name for your filter.
4. In the **Filter Scope** group box, under **Item type**, select the item type you want to filter, for example **Cable**.

NOTE If you are applying a filter to all the CBDs (by right-clicking on the **Cable Block Diagrams** folder) then only **Item type: Block Diagram** is available.

5. In the **Filter definition** group box, do the following:
 - a. From the **Property list** select the required property, for example **Cable Type (Cable)**.
 - b. From the **Operator** list select an operator, for example **=**.
 - c. From the **Value** list select a value, for example **Type A**.
6. To add another definition to your filter, do one of the following:
 - If you want to use the same **Item type**, from the **Logic list** select an operator, click **Add** and then enter the next filter definition.
 - If you want to add a filter for a different item type, then select the item type from the **Item type** list, and then enter the next filter definition.

TIP The **Operators** and **Values** available, vary depending on the property chosen from the **Property** list.

7. When you have finished adding filter definitions, click **OK**.

TIP To clear a filter from the CBD, select the CBD in the **Cable Block Diagrams** folder, right-click and from the shortcut menu select **Clear Filter**.

SECTION 19

Foundation Fieldbus and Profibus

SmartPlant Instrumentation supports the creation and maintenance of Foundation Fieldbus and Profibus systems. Note that whenever fieldbus is mentioned in the subsequent topics of the documentation, we refer to Foundation Fieldbus.

Foundation Fieldbus Design

Foundation Fieldbus technology has rapidly established itself as a viable process industry networking methodology. Of the different protocols available currently on the market, Foundation Fieldbus is the leading product in the process industry. The SmartPlant Instrumentation fieldbus solution is based on H1 Intrinsically Safe Low Speed, 31.25 kb/sec Fieldbus, and supports topologies such as star, trunk/drop, and daisy chain.

The software provides the following features for creating and managing fieldbus projects:

- Fieldbus device tags, which you create in the Instrument Index module, include fieldbus properties such as identification references, address, electrical properties, associated function blocks, and so forth.
- The **Fieldbus Segment Manager**, where you create and manage fieldbus instruments and associate them with appropriate segments. Also, you can generate a validation report for a selected segment.
- Fieldbus connection apparatus and fieldbus bricks, which you create in the Wiring module. You can quickly and easily create fieldbus junction boxes and device panels using the **Plug-and- Socket Box Wizard**.
- As you build segments, the software verifies that you are following basic connection rules. Later, you can generate reports to verify that you built the segments according to further connection rules, and according to the segment-wide parameter profiles that you applied to each segment.
- Segment wiring diagrams and Fieldbus Loop enhanced reports.

NOTE Whenever fieldbus is mentioned in the subsequent topics of the documentation, we refer to Foundation Fieldbus.

See Also

Profibus Design (on page 211)

Flow of Activities for Foundation Fieldbus Design

The following is a suggested flow of activities for Foundation Fieldbus design in SmartPlant Instrumentation. Note that you can carry out some of the procedures in a different order, depending on your preferences or requirements.

1. Define the Segment-Wide Parameters for the current site

Segment-wide parameters allow you to define default settings that serve as design rules and limitations for the various Foundation Fieldbus and Profibus PA segments. These parameters affect the function and performance of your fieldbus system in the following ways:

- If you exceed cable lengths, or hook too many devices to the same pair of wires, this can cause attenuation and reduction in the voltage reaching the instruments. These voltage drops can result in instrument malfunction and bandwidth problems.
- If you create a spur cable that is too long, the signal propagated along that spur cable can become attenuated and too weak for the DCS to read.
- If you create a segment whose total capacitance exceeds the maximum, you may reduce effective bandwidth.

For more information, see *Define Segment-Wide Parameters* (on page 170).

2. Define Foundation Fieldbus instrument type profiles

- Prior to creating new fieldbus instruments, you need to customize wiring reference items and then define appropriate instrument type profiles in the Instrument Index module so that new tag numbers acquire fieldbus properties. This results in automatic assignment of the required properties to the new tag numbers. For more information, see *Define Foundation Fieldbus and Profibus Instrument Type Profiles* (on page 176).

3. Add a new browser view

- In the Browser module, create a browser view with the required fields for the **Fieldbus Segment Manager**. This facilitates the assignment of tag numbers to segments. You can create multiple Browser views with different Style, Sort, and Filter functions. For example, by using the filter function you may create independent views for each segment, so that only tags of that segment will be displayed in its corresponding view. For more information, see *Add a New View*.

4. Create user-defined function blocks

- Since the software does not allow you to modify or delete the shipped function blocks, you can create your own function blocks which you can modify or delete as you see fit. The function blocks that you define here are available for the current site. You can define function blocks as multiple with or without execution time. Instruments can be associated with several function blocks of the same type if that function block is defined as Multiple. Using the **Execution Time** feature allows you to set the time for the function block execution. You set the number of function blocks and the execution time in the **Function Block - Instrument Type Association** dialog box. You can create a virtual tag and associate it with an existing function block. Only one virtual tag can be associated with a function block. For more information, see *Create a User-Defined Function Block* (on page 172).

5. Associate function blocks with instrument types

- Associating function blocks with instrument types enables you to instruct the software to create and associate a particular function block when creating a fieldbus instruments. When you create a fieldbus instrument tag, SmartPlant Instrumentation automatically assigns the function blocks associated with the instrument type that you select for the new tag. Note that you can associate a function block with as many instrument types as required. For more information, see *Associate Function Blocks with Instrument Types* (on page 173).
- 6. Create Foundation Fieldbus instruments**
- Create your fieldbus device tags based on the instrument types that you defined, and edit the tag number properties as required, including function block properties. The software allows you to create various types of instruments that can be associated with the Foundation Fieldbus system in your plant. In other words, you can create instruments that are compatible with Foundation Fieldbus, Profibus DP, and Profibus PA segments. You add new instruments in the Instrument Index module, exactly the same way that you create other tags. Once you define an instrument type for the new tag number and associate a function block with the new tag's instrument type, the new instrument automatically acquires all of the appropriate properties that you set for that tag number. You can then create virtual tags for this instrument. For more information, see *Create Foundation Fieldbus and Profibus Instruments* (on page 176).
- 7. Associate your function blocks with fieldbus instruments**
- This feature enables you to associate a function block with a particular instrument. You use this feature if for some reason, the current tag number has not been associated with any function block through the tag's instrument type or if you need to change some of the associations. Note that the association or any changes made to the association will affect the current tag number only. For more information, see *Associate Function Blocks with Instruments* (on page 179).
- 8. Create your fieldbus segments and associate them with fieldbus instruments**
- Now you are ready to create your fieldbus segments, associate the appropriate instruments with their respective segments, and create the necessary virtual tags. For more information, see the following:
 - *Create a Foundation Fieldbus Segment* (on page 182)
 - *Associate an Instrument with a Segment* (on page 183)
 - *Enable a Function Block for I/O Assignment* (on page 184)
- 9. Design your wiring and termination equipment**
- At this stage you need to create your fieldbus I/O cards and if you are going to use the plug-and-socket communication system, create your fieldbus plug-and-socket boxes. For more details, see the following:
 - *Create a Fieldbus I/O Card* (see "*Create a Foundation Fieldbus Segment*" on page 182)
 - *Fieldbus Plug-and- Socket Boxes* (see "*Fieldbus Plug-and-Socket Boxes (Panels)*" on page 186)
- 10. Add your home-run cables and spurs.**
- For details, see *Managing Fieldbus Cables Common Tasks* (see "*Fieldbus Cables Common Tasks*" on page 185).

11. Make the required connections and associations

- At this stage you are ready to make the required connections and associations. However, before you can associate a segment with a home-run cable or connect a home-run cable to a plug-and-socket box, you need to configure various connector types assign connector pins to wires. For more information, see *Making Connections and Associations for Fieldbus Common Tasks* (see "Making Connections and Associations for Fieldbus" on page 199).

12. Effect the I/O assignment

- For details, see *Effect Fieldbus I/O Assignment* (on page 206).

13. Generate fieldbus validation reports

- As you build the segments, and make connections and effect I/O assignments, the software verifies that you are following basic fieldbus rules. Later, you can generate reports to verify that you built the segments according to additional connection rules and according to the segment-wide parameter profiles that you applied to each segment. For details, see *Fieldbus Validation* (on page 207).

14. Generate a segment wiring report

- You can generate an enhanced segment wiring drawing to view the completed segment. For details, see *Generate a Segment Wiring Report* (on page 210).

15. Generate a specification

- In the Specifications module you can associate tags with fieldbus forms to generate specifications. For details, see *Principles of Generating Specifications*.
- If you encounter difficulty in activating a library form for fieldbus, you should restore the form. For details, see *Restore Library Forms*.
- Also, you can add function block custom fields to fieldbus specifications. For details, see *Add Function Block Custom Fields to a Fieldbus Spec*.

16. Generate a hook-up drawing

- You can also generate a hook-up drawing for your fieldbus instruments just like for conventional instruments. For details see *Hook-Up Drawing Generation* (on page 331).

SECTION 20

Preliminary Definitions for Foundation Fieldbus Common Tasks

The following tasks are used frequently when you make preliminary Foundation Fieldbus definitions.

Define Segment-Wide Parameters

Segment-wide parameters allow you to define default settings that serve as design rules and limitations for the various Foundation Fieldbus and Profibus PA segments. These parameters affect the function and performance of your fieldbus system in the following ways:

- If you exceed cable lengths, or hook too many devices to the same pair of wires, this can cause attenuation and reduction in the voltage reaching the instruments. These voltage drops can result in instrument malfunction and bandwidth problems.
- If you create a spur cable that is too long, the signal propagated along that spur cable can become attenuated and too weak for the DCS to read.
- If you create a segment whose total capacitance exceeds the maximum, you may reduce effective bandwidth.

For more information, see *Define Segment-Wide Parameters* (on page 170).

Create a User-Defined Function Block

You use this feature to add new user-defined function blocks, delete redundant ones, or modify the definition of existing user-defined function blocks. The function blocks that you define here are available for the current site. You can define function blocks as multiple with or without execution time. Instruments can be associated with several function blocks of the same type if that function block is defined as Multiple. Using the **Execution Time** feature allows you to set the time for the function block execution. You set the number of function blocks and the execution time in the **Function Block - Instrument Type Association** dialog box. You can create a virtual tag and associate it with an existing function block. Only one virtual tag can be associated with a function block.

For more information, see *Create a User-Defined Function Block* (on page 172).

Associate Function Blocks with Instrument Types

Associating function blocks with instrument types enables you to instruct the software to create and associate a particular function block when creating a fieldbus instruments. When you create a fieldbus instrument tag, SmartPlant Instrumentation automatically assigns the function blocks associated with the instrument type that you select for the new tag. Note that you can associate a function block with as many instrument types as required.

For more information, see *Associate Function Blocks with Instrument Types* (on page 173).

Manage the Function Blocks Supporting Table

This option shows you how to use this supporting table to add new user-defined function blocks to your instrument index, delete the redundant ones, or modify the definition of existing user-defined function blocks. You can also enable the **Multiple** and **Execution Time** features for any existing function block. For more information, see *Manage the Function Blocks Supporting Table* (on page 174).

Define Segment-Wide Parameters

1. Start the Wiring module.
2. On the main menu bar, click **Tables > Segment-Wide Parameter Profiles**.
3. In the **Segment-Wide Parameter Profiles** dialog box, click **New**.
4. In the **Segment-Wide Parameter Profiles Properties** dialog box, click the **General** tab, and do the following:
 - a. In the **Name** box, type a unique name for the profile.
 - b. In the **Description** box, type a description for the profile.
 - c. Select the profile as **Default**, or leave it unselected.

TIP If selected, when you add a new segment, this profile will be the default setting associated with the new segment.
5. In the **Maximum number of devices** group box, do the following:
 - a. **Per spur** — Type the maximum number of tag numbers that can be connected with one spur.
 - b. **Per segment** — Type the maximum number of tag numbers that can be associated with one segment.
 - c. **Per intrinsically safe (IS) segment** — Type the maximum number of tag numbers that can be associated with one intrinsically safe segment.
6. In the **Maximum number of assigned function blocks** group box, do the following:
 - a. **Per I/O card** — Type the maximum number of function blocks that can be associated with one I/O card.
 - b. **Per segment** — Type the maximum number of function blocks that can be associated with one segment.
7. In the **Electrical parameters** group box, do the following:
 - a. **Power supply** — Type the available voltage source value, in volts.
 - b. **Minimum receiver voltage** — Type the minimum receiver voltage value that can be read by a DCS, in volts.
 - c. **Allowable current per non IS segment** — Type the maximum allowable value for the current consumption in the non-intrinsic safety segments, in milliamperes.
 - d. **Allowable current per IS segment** — Type the maximum allowable value for the current consumption in the intrinsic safety segments, in milliamperes.
 - e. **Maximum capacitance** — Type the maximum allowable segment capacitance in the unit of measure selected at right.
8. Click the **Cable Lengths** tab.
9. In the **Recommended spur cable lengths** group box, define the rules for spur cables connecting instruments to home-run cables. Do the following:
 - a. Select the required unit of measure (meters or feet).
 - b. In the **Number of devices per segment** columns, type the number of devices per segment for which you want to set the spur lengths. Make sure that you do not exceed the number of devices that you defined in the **Maximum number of devices** group box.
 - c. In the **Number of devices per spur** columns, type the maximum allowable length of the spur cables for each definition.
 - d. You can click **New** to add a row, or **Delete** to delete the row marked by .
 - e. See *Spur Cable Lengths Example* (on page 171).

10. Click **OK** to confirm your creation of this profile and return to the **Segment-Wide Parameter Profiles** dialog box.

TIP You can return to Step 3 to create an additional segment-wide parameter profile.

11. Click **OK** to close the **Segment- Wide Parameter Profiles** dialog box and return to the main window of the **Wiring** module.

See Also

Profibus Design (on page 211)

Fieldbus Validation (on page 207)

Spur Cable Lengths Example

The following example shows the rules that are set for the lengths of individual spur cables connecting a junction box to instruments in a variety of fieldbus topologies. For example, as shown in row 3, where 15 to 18 devices are associated per segment, and one instrument is associated per spur, the cable can be up to 60 meters long. If we allow 2 instruments per spur with the same number of associated devices per segment, each cable can run up to 30 meters, and so forth.

Recommended spur cable lengths

Units: ☒ Meters ☐ Feet

Number of devices per segment		Number of devices per spur			
		1	2	3	4
1	12	120	90	60	30
13	14	90	60	30	1
15	18	60	30	1	1
19	24	30	1	1	1
25	32	1	1	1	1

In this example, the recommended spur cable lengths are:

- 120 meters for 1 to 12 devices per segment and one device per spur;
- 90 meters for 1 to 12 devices per segment and two devices per spur;
- 60 meters for 1 to 12 devices per segment and three devices per spur;
- 30 meters for 1 to 12 devices per segment and four devices per spur;
- 90 meters for 13 to 14 devices per segment and one device per spur; and so forth.

Working with Function Blocks

Function blocks are smart devices that hold control software on fieldbus instruments. These smart devices or microprocessors enable Foundation Fieldbus and Profibus PA instruments to perform control functions independently of a DCS. There are many types of function blocks. Each function block has a specific purpose. SmartPlant Instrumentation provides a set of the most common function blocks, with properties defined by Fieldbus Foundation. You cannot delete these shipped function blocks or rename them, but you can modify their definitions if needed. You can, however, create your own user-defined function blocks for both Fieldbus Foundation and Profibus PA instruments. You can modify and delete these function blocks as you need.

Function blocks are associated with particular instrument types. When creating a new instrument based on an instrument type that has been associated with a function block, the software creates that instrument and automatically associates it with the function block have been set for pertinent instrument type. The software allows you to associate multiple function blocks with a single instrument type. For example, a basic flow transmitter will have one AI (analog input) block associated with it. However, a similar instrument may be used to measure also pressure and temperature (required for gas flow compensation), and may be further equipped with one or more alarm functions. An AI function block is required for each additional process measurement and an AA (analog alarm) block for each alarm function. You can also associate existing instruments with function blocks, or modify associations as needed.

NOTES

- You cannot create virtual tags for an instrument that is not associated with at least one function block.
- A number of Foundation Fieldbus and Profibus PA function blocks are shipped with the software. You cannot add, delete or modify any of them. You can see all the available function block types on the **Function Block – Instrument Type Association** dialog box (for details, see *Associate Function Blocks with Instrument Types* (on page 173)).

See Also

Create Foundation Fieldbus and Profibus Instruments (on page 176)

Associate Function Blocks with Instruments (on page 179)

Profibus Design (on page 211)

Create a User-Defined Function Block

1. Start the Instrument Index module.
2. On the Instrument Index module menu bar click **Tables > Function Blocks**.
3. In the **Function Blocks** dialog box, do one of the following:
 - To add a new function block, click **New**.
 - To modify the definition of a highlighted function block, click the field that want to modify.
4. In the **Function Block** field, type the function block name.
5. In the **Description** field, type a short description as appropriate.
6. To define the function block as a multiple function block, select **Multiple**.
TIP This will enable you to associate multiple copies of a given functional block with an instrument type or a specific instrument.
7. To enable entry of an execution time value, select **Execution Time**.
8. Click **OK**.

Associate Function Blocks with Instrument Types

1. Start the Instrument Index module.
2. On the Instrument Index module menu bar click **Tables > Instrument Types**.
3. In the **Instrument Types** dialog box, select a process function type and an instrument type.
4. Click **Profile**.
5. On the **Instrument Type Profile** dialog box, on the **General** tab, select the **Include I/O type** check box.
6. From the **Include I/O type** list, select one of the following:
 - **FieldbusFF Foundation Fieldbus**
 - **Profibus PA**
7. In the **Function Block - Instrument Type Association** dialog box, to assign a function block to an instrument type, do one of the following:
 - In the **Unassociated function blocks** data window, select the function block that you want to associate, and click **Associate**.
 - Drag the required function block from the **Unassociated function blocks** data window to the **Associated function blocks** data window.
8. If the required function block is not available in the **Function Block - Instrument Type Association** dialog box, add or modify the function blocks in the **Function Blocks** dialog box. For details, see *Manage the Functions Blocks Supporting Table* (see "Manage the Function Blocks Supporting Table" on page 174).
9. If you customized a given function block in the **Function Blocks** dialog box, do the following as needed:
10. To set the maximum number of instruments that can be associated with the current function block, in the **Associated function blocks** data window, under **Multiple**, type the value.
11. To set the execution time, under **Execution Time**, type the time value (in milliseconds).
12. Click **OK**.

See Also

Define Foundation Fieldbus and Profibus Instrument Type Profiles (on page 176)

Associate Function Blocks with Instruments (on page 179)

Profibus Design Common Tasks (on page 213)

Manage the Function Blocks Supporting Table

1. On the Instrument Index module menu bar, click **Tables > Function Blocks**.
2. Click **New** to append a new data row or click in a field of a selected row to edit the data.
3. Type the function block values in the appropriate fields as follows:
 - **Function Block** — Enter the required function block name.
 - **Description** — Type a description for the function block if required.
 - **Multiple** — Select the check box to define this function block as a multiple function block. This will enable you to create multiple function blocks of the same type when adding or editing a instrument associated with this function block.
 - **Execution Time** — Select the check box to enable execution time definition for the current function block. You will be able to set the execution time when associating this function block with an instrument type or an instrument tag.
4. Click **OK**.

NOTE You cannot delete or rename shipped function blocks.

SECTION 21

Fieldbus Instruments Common Tasks

The following tasks are used frequently when you manage instruments for your Foundation Fieldbus and Profibus systems.

Define Foundation Fieldbus and Profibus Instrument Type Profiles

Prior to creating new fieldbus instruments, you need to customize wiring reference items and then define appropriate instrument type profiles in the Instrument Index module so that new tag numbers acquire fieldbus properties. This results in automatic assignment of the required properties to the new tag numbers. For more information, see *Define Foundation Fieldbus and Profibus Instrument Type Profiles* (on page 176).

Create Foundation Fieldbus and Profibus Instruments

Create fieldbus device tags based on the instrument types that you defined, and edit the tag number properties as required, including function block properties.

The software allows you to create various types of instruments that can be associated with the Foundation Fieldbus or Profibus system in your plant. In other words, you can create instruments that are compatible with Foundation Fieldbus, Profibus DP, and Profibus PA segments. You add new instruments in the Instrument Index module, exactly the same way that you create other tags. Once you define an instrument type for the new tag number and associate a function block with the new tag's instrument type, the new instrument automatically acquires all of the appropriate properties that you set for that tag number. You can then create virtual tags for this instrument.

For more information, see *Create Foundation Fieldbus and Profibus Instruments* (on page 176).

Associate Function Blocks with Instruments

This feature enables you to associate a function block with a particular instrument. You use this feature if for some reason, the current tag number has not been associated with any function block through the tag's instrument type or if you need to change some of the associations. Note that the association or any changes made to the association will affect the current tag number only. For more information, see *Associate Function Blocks with Instruments* (on page 179).

Working with the Fieldbus Tag Numbers Browser

The **Fieldbus Segment Manager** provides for the creation and management of fieldbus instrument tags and their association with appropriate segments. Also, you can generate a validation summary report for a selected segment.

The manager displays all the instruments that exist in your fieldbus system. The instrument properties are displayed in various columns, such as **Tag Number**, **Loop Number**, **Service**, **Segment Name**, and so forth. The columns, their sequence, and row sorting in this pane depend on the Browser view currently selected in the **Browser Manager**. The default display is the **New Fieldbus Tag Number List** view in the **Browser Manager**.

To open the **Fieldbus Segment Manager** browser, do the following:

1. In the **Domain Explorer**, expand the **Fieldbus Segments** folder.
2. Right-click a segment.
3. On the shortcut menu, click **Actions > Fieldbus Segment Manager**.

Define Foundation Fieldbus and Profibus Instrument Type Profiles

1. On the Instrument Index module menu bar, click **Tables > Instrument Types**.
2. In the **Instrument Types** dialog box, from the **Process function** list, select an appropriate process function.
3. Do one of the following:
 - Click **New** to create a new instrument type. For details, see Define an Instrument Type.
 - In the data window, select an existing instrument type.
4. Click **Profile**.
5. In the **Instrument Type Profile** dialog box, on the **General** tab, under **Specification**, select an appropriate fieldbus specification.
6. Select **Include I/O type**, and then from the list, select one of the following:
 - **Foundation Fieldbus**
 - **Profibus DP**
 - **Profibus PA**
7. On the **Wiring and Control System** tab, select the appropriate reference device panel and reference cable. For details, see Define a Wiring and Controls System Instrument Type Profile.
8. Click the **Fieldbus** tab.

TIP This tab is available only when you select **Foundation Fieldbus** from the **Include I/O type** list. For Profibus instruments, click **OK** to complete the procedure.
9. On the **Fieldbus** tab, select the **Include fieldbus** check box.
10. Type a fieldbus tag name.
11. Type a fieldbus device address.
12. Type a device identification.
13. Type a DC consumption value in milliamperes.
14. Enter a capacitance setting and select the required unit of measure.
15. Enter a minimum transmit level value.
16. Set a default operating voltage value.
17. Select the **Backup link master** check box to set new fieldbus instruments as backup link masters that will take over control functions for the loop in case of DCS failure.
18. Click **Function Block** to associate function blocks with instrument types. For details, see *Associate Function Blocks with Instrument Types* (on page 173).
19. Click **OK** in the **Instrument Type Profile** dialog box.

See Also

Profibus Design Common Tasks (on page 213)

Create Foundation Fieldbus and Profibus Instruments

NOTE The current procedure allows you to create fieldbus and profibus instruments from scratch. If you already have such instruments, you can create new ones by duplicating the existing instruments. In this case, the new fieldbus and profibus tags inherit the source segments.

1. Press F7 to open in the **Domain Explorer**.

2. Expand the plant hierarchy to display the **Instruments** and **Loops** folders.
3. Do one of the following:
 - To create an instrument unassociated with a loop number, right-click the **Instruments** folder, and then click **New > Instrument**.
 - To create an instrument that is associated with a specific loop number, expand the **Loops** folder, right-click a loop, and then click **New > Instrument**.

TIP To create new fieldbus tag numbers from the **Fieldbus Tag Numbers** browser, right-click somewhere in the browser, and then click **New Fieldbus Tag**. Continue as described below.

4. On the **New Tag Number** dialog box, from the **Tag class** list select one of the following:
 - **Foundation Fieldbus** — an instrument that can be associated with a Foundation Fieldbus segment.
 - **Profibus DP** — an instrument that can be associated with a Profibus DP segment.
 - **Profibus PA** — an instrument that can be associated with a Profibus PA segment.
5. Under **Tag number**, type the name of the new tag number, and then, click **OK**.
6. On the **Select Instrument Type** dialog box, select the instrument type you want to associate with the current tag number, and then, click **OK**.
7. Click **OK**.

TIPS

- If the tag number does not correspond to an existing loop name, SmartPlant Instrumentation prompts you to enter a loop name based on the tag number you have entered. If the loop identifier already matches an existing loop, the software will automatically associate the new tag number with the loop.
 - If the loop convention includes the loop function parameter, the prompt will always appear. In this case, you have to complete the loop number. If a profile exists for the selected instrument type, any new tags for that instrument type will be created with the selected reference items.
 - For DeltaV compatibility, you must select an instrument type that you associated with a DeltaV device type during the downloading process. To display the **Associate Foundation Fieldbus Device Types with Instrument Type** supporting table of associations that you built among DeltaV device types, instrument manufacturers, and SmartPlant Instrumentation instrument types, see Associate Device Types for DCS Vendors.
8. On the **Loop Name** dialog box, do one of the following:
 - Type the loop number that the new tag is associated with.
 - Accept the displayed loop number.
 - Click **Cancel** to create the tag number without a loop association. Note that if a loop with the same name exists, the tag number is automatically associated with it, without creating a new loop.
 9. Click **OK** to create the loop number.
 10. On the **Loop Number Properties** dialog box, accept the loop number properties or modify them as you require and then click **OK**.
 11. On the **Tag Number Properties** dialog box, on the **General** tab, enter the tag number attributes that you require.

TIP For DeltaV compatibility, make sure that the value that you select from the **Manufacturer** list is among the acceptable values for the instrument type that you selected for the current tag. For details, see Crucial Fields for the DeltaV Interface.

12. On the **Fieldbus** tab, to define the fieldbus properties for the new instrument, type the fieldbus tag name.
13. Type the field device address if necessary.
14. Type the device ID if necessary.
15. If the new tag functions as a backup that takes over DCS functions in case of DCS failure in the current segment, select **Backup link master**.
16. Type the manufacturer's fieldbus device revision number if necessary.

TIPS

- If your DeltaV interface preferences are set for fieldbus mode, the **Fieldbus device revision** field is a list, from which you must select a revision.
- SmartPlant Instrumentation uses the values that you enter below to validate fieldbus segments. For details, see *Generate a Validation Report for One Segment* (on page 209).

17. Type the DC current consumption of the current instrument, for example, 35 mA.
18. Type the capacitance and the capacitance unit of measure, for example 5 pF.
19. Type the minimum transmit level of the current instrument, for example, 2.5 volts.
20. Type the operating voltage range, for example, 9 - 35 volts.
21. To modify the function block association of the new instrument, in the **Function blocks association** group box, under **Unassociated function blocks**, select a function block that you want to associate with the instrument, and click **Associate**.

TIP If your DeltaV interface preferences are set for fieldbus mode, your selection of manufacturer and of fieldbus device revision determine function block association for the current tag.

22. To change the number of copies of the function block, in the **Associated function blocks** data window, under **Multiple**, type the value.

TIP This option is available only if you enabled it in the **Function Blocks** dialog box.

23. To set the execution time, under **Execution Time**, type the time value (in milliseconds).

TIP This option is available only if you enabled it in the **Function Blocks** dialog box.

24. To enter power supply properties, see *Enter Power Supply Data for Panels and Instrument Tags* (on page 45).

25. Click **OK**.

IMPORTANT If you are creating items that must be compatible with Emerson DeltaV, see Crucial Fields for the DeltaV Interface.

See Also

Working with Function Blocks (on page 172)

Associate Function Blocks with Instrument Types (on page 173)

Profibus Design Common Tasks (on page 213)

Associate Function Blocks with Instruments

1. In the **Domain Explorer**, expand the **Instruments** folder and then right-click a tag number.
2. On the shortcut menu, click **Properties**.
3. On the **Tag Number Properties** dialog box, click the **Fieldbus** tab.
4. Under **Unassociated function blocks**, select the function block that you want to associate, and click **Associate**.
5. To set the maximum number of instruments that can be associated with the current function block, in the **Associated function blocks** data window, under **Multiple**, type the value.
TIP The **Multiple** field is editable only if it was enabled on the **Function Blocks** dialog box.
6. To set the execution time, under **Execution Time**, type the time value in milliseconds.
TIP The **Execution Time** field is editable only if it was enabled on the **Function Blocks** dialog box.
7. Click **OK**.

See Also

Define Foundation Fieldbus and Profibus Instrument Type Profiles (on page 176)

Working with Function Blocks (on page 172)

Create a User-Defined Function Block (on page 172)

Associate Function Blocks with Instrument Types (on page 173)

Profibus Design Common Tasks (on page 213)

SECTION 22

Fieldbus and Profibus Segments Common Tasks

The following tasks are used frequently when you manage segments for your Foundation Fieldbus and Profibus systems.

Create a Foundation Fieldbus Segment

A Foundation Fieldbus segment is a group of devices physically connected by a single pair of wires to a host control device. This procedure explains how to add a new segment to your fieldbus system. For more information, see *Create a Foundation Fieldbus Segment* (on page 182).

Edit the Properties of a Foundation Fieldbus Segment

This feature allows to edit the properties of a Foundation Fieldbus segment. You can rename a segment, select another segment-wide parameter profile, and set the segment as intrinsically safe. For more information, see *Edit the Properties of a Foundation Fieldbus Segment* (on page 182).

Delete a Fieldbus Segment

Use this feature to delete a segment. For more information, see *Delete a Fieldbus Segment* (on page 184).

Associate an Instrument with a Segment

This feature enables you to associate a segment with a particular instrument. You use this feature if for some reason, the current tag number has not been associated with any segment or if you need to change some of the associations. Note that the association or any changes made to the association will affect the current tag number only. For more information, see *Associate an Instrument with a Segment* (on page 183).

Dissociate an Instrument from a Segment

You use this procedure when you need to dissociate a particular instrument from its segment. For more information, see *Dissociate an Instrument from a Segment* (on page 183).

Change Instrument / Segment Association

You use this procedure when you need to dissociate a particular instrument from its segment. For more information, see *Change Instrument / Segment Association* (on page 183).

Enable a Function Block for I/O Assignment

This procedure shows how to enable a function block for I/O assignment. You can do this after associating an instrument with a segment. Note that without enabling function blocks, segment I/O assignment cannot be effected. For more information, see *Enable a Function Block for I/O Assignment* (on page 184).

Change the View in the Fieldbus Tag Numbers Browser

This option allows you to change the current view of the **Fieldbus Tag Numbers** browser. The view, (that is, the available fields) depends on the current view in the **Fieldbus Tag Number List** browser. For more information, see *Change the View in the Fieldbus Tag Numbers Browser* (see "Change the Fieldbus Tag Number List Browser View for Segments" on page 184).

Create a Foundation Fieldbus Segment

1. In the **Domain Explorer**, right-click the **Fieldbus Segments** folder and from the shortcut menu, click **New > Fieldbus Segment**.
2. On the **Foundation Fieldbus Segment Properties** dialog box, under **Segment**, type a unique name for the new segment.
3. Select a profile from the **Segment-wide parameter profile** list. If necessary, click  to open the **Segment-Wide Parameter Profile** dialog box to edit or define a new segment-wide parameter profile. For details, see *Define Segment-Wide Parameters* (on page 170).
4. Do one of the following:
 - Select **Intrinsically safe** for this segment. SmartPlant Instrumentation warns you if any instruments currently associated with this segment are not intrinsically safe. The software also warns you if an instrument you try to associate with this segment in the future is not intrinsically safe.
 - Clear **Intrinsically safe**. SmartPlant Instrumentation warns you if any instruments currently associated with this segment are intrinsically safe. The software will also warn you if an instrument you try to associate with this segment in the future is intrinsically safe.
5. Click **OK** to create the new segment.

See Also

Connect a Home-Run Cable to a Plug-and-Socket Box (on page 202)

Edit the Properties of a Foundation Fieldbus Segment

1. In the **Domain Explorer**, double-click the **Fieldbus Segments** folder.
2. Right-click a segment and then on the shortcut menu, click **Properties**.
3. In the **Foundation Fieldbus Segment Properties** dialog box, under **Segment**, type a unique name for the new segment.
4. Select a profile from the **Segment-wide parameter profile** list. If necessary, click  to open the **Segment-Wide Parameter Profile** dialog box to edit or define a new segment-wide parameter profile. For details, see *Define Segment-Wide Parameters* (on page 170).
5. Do one of the following:
 - Select **Intrinsically safe** for this segment. SmartPlant Instrumentation warns you if any instruments currently associated with this segment are not intrinsically safe. The software also warns you if an instrument you try to associate with this segment in the future is not intrinsically safe.
 - Clear **Intrinsically safe**. SmartPlant Instrumentation warns you if any instruments currently associated with this segment are intrinsically safe. The software will also warn you if an instrument you try to associate with this segment in the future is intrinsically safe.
6. Click **OK** to create the new segment.

Associate an Instrument with a Segment

1. On the Wiring module menu bar, click **Actions > Fieldbus Segment Manager**.
2. In the **Fieldbus Segment Manager**, select an appropriate tag number.
3. Drag the highlighted instrument to the required segment in the **Fieldbus Segments** folder of the **Domain Explorer**.

NOTES

- The associated instrument appears under the selected segment in the **Fieldbus Segments** folder of the **Domain Explorer**.
- A fieldbus instrument can be associated only with one fieldbus segment. However, you can associate multiple tag numbers with a given fieldbus segment.

See Also

Profibus Design Common Tasks (on page 213)

Dissociate an Instrument from a Segment

1. In the **Domain Explorer**, double-click the **Fieldbus Segments** folder.
2. Expand the hierarchy of a segment.
3. Right-click an instrument and then on the shortcut menu, click **Actions > Dissociate Item**.

NOTE The **Segment Name** field in the **Instruments** pane displays **** for dissociated instruments.

See Also

Profibus Design Common Tasks (on page 213)

Change Instrument / Segment Association

1. On the Wiring module menu bar, click **Actions > Fieldbus Segment Manager**.
2. In the **Domain Explorer**, double-click the **Fieldbus Segments** folder and expand the appropriate segments.
3. Right-click an instrument tag, and then, on the shortcut menu, click **Actions > Dissociate Item from Segment**.
4. In the **Fieldbus Segment Manager**, select the dissociated tag and drag it to the target segment in the **Fieldbus Segments** folder.

NOTES

- You cannot change the segment association of a connected instrument before disconnecting its wiring.
- The **Segment Name** field in the **Fieldbus Segment Manager** displays **** for dissociated instruments.

See Also

Profibus Design Common Tasks (on page 213)

Delete a Fieldbus Segment

1. In the **Domain Explorer**, double-click the **Fieldbus Segments** folder.
 2. Select a segment you want to delete, and do one of the following:
 - Press the **Delete** key.
 - Right-click a segment and then on the shortcut menu, click **Delete**.
 3. Click **Yes** to confirm the segment deletion.
- NOTE** After deleting a segment, the software dissociates all the associated instruments and deletes the relevant virtual tags.

Enable a Function Block for I/O Assignment

1. In the **Domain Explorer**, under **Fieldbus Segments**, expand a segment  to display its associated instruments.
2. Right-click the desired function block  under the highlighted tag number.
3. On the shortcut menu, click **Actions > Enable**.

Change the Fieldbus Tag Number List Browser View for Segments

1. In the **Instrument Index** Browser Group of the **Browser Manager**, define a view under the **Fieldbus Tag Number List Browser**, for use in the **Fieldbus Segment Manager**. For details of how to define a view, see *Add a New View*.
2. In the **Fieldbus Segment Manager**, do one of the following:
 - Click **Actions > Select Browser View for Segments**.
 - Click .
3. On the **Segment Views** dialog box, select the desired view and then click **OK**.

SECTION 23

Fieldbus Cables Common Tasks

The following tasks are used frequently when you manage fieldbus cables for your Foundation Fieldbus and Profibus systems.

Create a Home-Run Cable

A home-run cable is the main communication highway between devices on a fieldbus network. A home-run cable serves the spurs in a given segment, and has a terminator at either end. For more information, see *Create a Home-Run Cable* (on page 185).

Create a Spur Cable

A spur is a part of a segment that drops off from the main segment (home-run) trunk. A spur may contain one or more instruments and cables that connect them to the home-run segment. For more information, see *Create a Spur Cable* (on page 186).

Create a Home-Run Cable

1. Press F7 to open the **Domain Explorer**.
2. Right-click the **Cables** folder and then on the shortcut menu, click **New > Home-Run Cable**.
3. In the **Cable Configuration** dialog box, do one of the following:
 - From the **Cable configuration** list, select an appropriate cable configuration (for example, Fieldbus cable).
 - Define a new cable configuration. For details, see *Define a Cable Configuration* (on page 26).
4. Click **Create**.
5. In the **Cable Properties** dialog box, under **Cable**, type a unique name.
TIP In the **Cable Properties** dialog box, make sure that you select **Fieldbus home-run** from the **Cable class** list.
6. Complete the creation of the home-run cable using the general procedure for creating a new cable. For details, see *Create a Cable* (on page 28).

See Also

Foundation Fieldbus Design (on page 165)

Associate a Segment with a Home-Run Cable (on page 203)

Profibus Design Common Tasks (on page 213)

Create a Spur Cable

1. Press F7 to open the **Domain Explorer**.
2. Right-click the **Cables** folder and then on the shortcut menu, click **New > Cable**.
3. In the **Cable Configuration** dialog box, do one of the following:
 - From the **Cable configuration** list, select an appropriate cable configuration (for example, Fieldbus cable).
 - Define a new cable configuration. For details, see *Define a Cable Configuration* (on page 26).
4. Click **Create**.
5. In the **Cable Properties** dialog box, under **Cable**, type a unique name.
TIP In the **Cable Properties** dialog box, make sure that you select **Conventional** from the **Cable class** list.
6. Complete the creation of the home-run cable using the general procedure for creating a new cable. For details, see *Create a Cable* (on page 28).

See Also

Foundation Fieldbus Design (on page 165)

Profibus Design Common Tasks (on page 213)

Fieldbus Plug-and-Socket Boxes (Panels)

You need plug-and-socket boxes to connect Foundation Fieldbus instruments to a spur or a home-run cable that has a plug-and-socket connection.

SmartPlant Instrumentation supports active plug-and-socket junction boxes, that is, plug-and-socket junction boxes that require power supply. You can define the current consumption, open circuit voltage, and short-circuit protection for an active plug-and-socket junction box. When generating a segment validation report, the software takes these values into account when calculating the total segment current consumption and voltage drop for a particular segment.

You use the Plug-and-Socket Box Wizard to create the required plug-and-socket boxes or reconfigure the existing ones.

Plug-and-socket boxes can be device panels or junction boxes.

SECTION 24

Managing Fieldbus Panels (Plug-and-Socket Boxes) Common Tasks

The following tasks are used frequently when you manage panels for your Foundation Fieldbus and Profibus systems.

You can create new panels either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

Create a Plug-and-Socket Device Panel

This procedure explains how to create a new plug-and-socket device panel. For more information, see *Create a Plug-and- Socket Device Panel* (see "Create a Plug-and-Socket Device Panel" on page 187).

Edit the Properties of a Plug-and-Socket Device Panel

This procedure explains how to modify the properties of a plug-and-socket device panel. For more information, see *Edit the Properties of a Plug-and-Socket Device Panel* (see "Create a Plug-and-Socket Device Panel" on page 187).

Create a Plug-and-Socket Junction Box

This feature allows to create a plug-and-socket junction box. For more information, see *Create a Plug-and-Socket Junction Box* (on page 188).

Edit the Properties of a Plug-and-Socket Junction Box

This feature explains how to modify the properties of a plug- and-socket junction box. For more information, see *Edit the Properties of a Plug-and-Socket Junction Box* (on page 189).

Create a Plug-and-Socket Device Panel

This procedure explains how to create a new plug-and-socket device panel.

1. Press F7 to open the **Domain Explorer**.
2. Do one of the following:
 - Double-click the **Panels by Location** folder, right-click a location and then on the shortcut menu click **New > Device Panel (Plug- and-Socket)**.
 - Double-click the **Panels by Category** folder, right-click the **Device Panels** folder and then on the shortcut menu click **New > Device Panel (Plug-and-Socket)**.

3. In the **Plug-and-Socket Box** wizard, follow all the instructions carefully and then click **Next** until you complete the wizard.

NOTES

- You can click  next to a list arrow to access the appropriate supporting table. This allows you to add, edit, or delete select list items.
- When defining the plug-and- socket box connectors, under **Connector type definition display**, SmartPlant Instrumentation displays the connector sequence, the appropriate pin number, and the pin polarities according to the configuration of the connector than you selected.

See Also

Profibus Design Common Tasks (on page 213)

Create a Plug-and-Socket Junction Box

1. Press F7 to open the **Domain Explorer**.
2. Do one of the following:
 - Double-click the **Panels by Location** folder, right-click a location and then on the shortcut menu click **New > Device Panel (Plug- and-Socket)**.
 - Double-click the **Panels by Category** folder, right-click the **Device Panels** folder and then on the shortcut menu click **New > Device Panel (Plug-and-Socket)**.
3. In the **Plug-and-Socket Box** wizard, follow all the instructions carefully and then click **Next** until you complete the wizard.

NOTES

- You can click  next to a list arrow to access the appropriate supporting table. This allows you to add, edit, or delete select list items.
- When defining the plug-and- socket box connectors, under **Connector type definition display**, SmartPlant Instrumentation displays the connector sequence, the appropriate pin number, and the pin polarities according to the configuration of the connector than you selected.

See Also

Profibus Design Common Tasks (on page 213)

Edit the Properties of a Plug-and-Socket Device Panel

1. Press F7 to open the **Domain Explorer**.
2. Do one of the following:
 - Double-click the **Panels by Location** folder, right-click a device panel and then on the shortcut menu click **Properties**.
 - Double-click the **Panels by Category** folder, right-click a device panel and then on the shortcut menu click **Properties**.
3. In the **Plug-and-Socket Box** wizard, follow all the instructions carefully and click **Next** until you complete the wizard.

NOTES

- The software does not allow you to modify port and connector configurations of an existing plug-and-socket device panel.

- You can click  next to a list arrow to access the appropriate supporting table. This allows you to add, edit, or delete select list items.
- When defining the plug-and- socket box connectors, under **Connector type definition display**, SmartPlant Instrumentation displays the connector sequence, the appropriate pin number, and the pin polarities according to the configuration of the connector than you selected.

See Also

Profibus Design Common Tasks (on page 213)

Edit the Properties of a Plug-and-Socket Junction Box

1. Press F7 to open the **Domain Explorer**.
2. Do one of the following:
 - Double-click the **Panels by Location** folder, right-click a junction box and then on the shortcut menu click **Properties**.
 - Double-click the **Panels by Category** folder, right-click a junction box and then on the shortcut menu click **Properties**.
3. In the **Plug-and-Socket Box** wizard, follow all the instructions carefully and click **Next** until you complete the wizard.

NOTES

- The software does not allow you to modify port and connector configurations of an existing plug-and-socket junction box.
- You can click  next to a list arrow to access the appropriate supporting table. This allows you to add, edit, or delete select list items.
- When defining the plug-and- socket box connectors, under **Connector type definition display**, SmartPlant Instrumentation displays the connector sequence, the appropriate pin number, and the pin polarities according to the configuration of the connector than you selected.

See Also

Profibus Design Common Tasks (on page 213)

Working with Terminators

Terminators are used to terminate and provide impedance compensation at a segment end. Two terminators are needed for each segment — one on the DCS/PLC terminal side (beginning of the segment) and the other on its other end.

There are two types of terminators — internal terminators (in relation to a terminating block) and external (fitted on as a separate accessory).

You can create internal or external terminators as required, whether it is a terminal apparatus strip or a plug-and-socket termination block.

You create internal terminators using the configuration tools when creating plug-and-socket boxes or termination blocks. External terminators are added to the termination block (an apparatus or a plug-and-socket box) during the wiring design phase.

See Also

Profibus Design Common Tasks (on page 213)

SECTION 25

Managing Terminators Common Tasks

The following tasks are used frequently when you manage terminators for your Foundation Fieldbus and Profibus systems.

Create an External Terminator in a Plug- and-Socket Box

This feature enables you to create an external terminator in an existing plug-and-socket box. For more information, see *Create an External Terminator in a Plug-and-Socket Box* (on page 192).

Create an External Terminator on a Terminal Strip

You can place an external terminator on a terminal in an apparatus or a conventional terminal strip, i.e., a junction box, marshaling rack, and so forth. For more information, see *Create an External Terminator on a Terminal Strip* (on page 192).

Move an External Terminator to Another Port in a Plug-and-Socket Box

Usually, an external terminator is added when you create a plug-and-socket box. You can disconnect a terminator or move it to another port in a plug-and-socket box if required. For more information, see *Move an External Terminator to Another Port in a Plug-and-Socket Box* (on page 193).

Create an Internal Terminator

Internal terminators are usually added when creating plug- and-socket boxes and termination blocks. You can also add an internal terminator to an existing plug-and-socket box or termination block if it has not been done during the creation process. If a plug-and-socket box does not contain an internal terminator, edit the properties of the required box in the **Plug-and-Socket Box** wizard. For more information, see *Create an Internal Terminator* (on page 192).

Terminator (Plug-and-Socket Box) — Segment Association

After creating an external terminator, you need to associate it with an existing segment. This option enables you to associate an external terminator with an existing segment where the terminator is located in a plug-and-socket box. For more information, see *Terminator (Plug-and-Socket Box) - Segment Association* (on page 193).

Terminator (Terminal Strip) — Segment Association

After creating an external terminator, you need to associate it with an existing segment. This option enables you to associate an external terminator with an existing segment where the terminator is located on a terminal strip. For more information, see *Terminator (Terminal Strip) - Segment Association* (on page 193).

Create an External Terminator in a Plug-and-Socket Box

1. In the **Domain Explorer** select a plug-and-socket box.
2. Right-click the selected plug-and-socket box and then on the shortcut menu, click **Actions > Connection**.
3. In the **Plug-and-Socket Box Connection** window, right-click the required port and then on the shortcut menu, click **New Terminator**.
4. In the **Plug-and-Socket Box Terminator** dialog box, make sure the correct port is displayed in the **Port** list. Select the required port if needed.
5. Click **Create** to add the new terminator and close the dialog box.

NOTE You can now associate this terminator with a segment. Right-click the selected terminator and then on the shortcut menu, click **Associate Segment**.

See Also

Profibus Design Common Tasks (on page 213)

Create an External Terminator on a Terminal Strip

1. In the **Domain Explorer**, select a panel.
2. Right-click the selected panel and then on the shortcut menu, click **Actions > Connection**.
3. In the **Connection** window, make sure that the required terminal strip is selected in the **Terminal Strip** list.
4. Right-click the required terminal and then on the shortcut menu, click **New Terminator**.
5. In the **New Terminator** dialog box, make sure the correct first terminal is displayed in the **First terminal** list.
Select the required first terminal if needed.
6. Select an appropriate terminal side.
7. Enter the required number of terminals that you want to terminate.
8. Click **Create** to add the new terminator and close the dialog box.
The new terminator appears in pink next to the selected terminal. You can now associate the terminal with a segment.

See Also

Profibus Design Common Tasks (on page 213)

Create an Internal Terminator

1. In the **Domain Explorer**, select a plug-and-socket box.
2. Right-click the selected plug-and-socket box and then on the shortcut menu, click **Properties**.
3. In the **Plug-and-Socket Box** wizard, click **Next** twice till you open the **Modify the Plug-and-Socket Layout** page in the **Plug-and-Socket Box** wizard.
4. Select the **Internal Terminator** check box and click **Next**.
5. Continue configuring the plug- and-socket box as needed.

See Also

Profibus Design Common Tasks (on page 213)

Move an External Terminator to Another Port in a Plug-and-Socket Box

1. In the **Domain Explorer**, select a plug-and-socket box.
2. Right-click the selected plug-and-socket box and then on the shortcut menu, click **Actions > Connection**.
3. In the **Plug-and-Socket Box Connection** window, right-click the required port and then on the shortcut menu, click **Disconnect Selected Items**.
4. Right-click the required port and on the shortcut menu, click **New Terminator**.
5. In the **Plug-and-Socket Box Terminator** dialog box, make sure the correct port is displayed in the **Port** list. Select the required port if needed.
6. Click **Create** to add the new terminator and close the dialog box.

NOTE You can now associate this terminator with a segment. Right-click the selected terminator and then on the shortcut menu, click **Associate Segment**.

See Also

Profibus Design Common Tasks (on page 213)

Terminator (Plug-and-Socket Box) - Segment Association

1. In the **Domain Explorer**, select a plug-and-socket box.
2. Right-click the selected plug-and-socket box and then on the shortcut menu, click **Actions > Connection**.
3. In the **Plug-and-Socket Box Connection** window, click the required connected home-run cable to select it.
4. Right-click the highlighted home-run cable and on the shortcut menu, click **Associate Segment** to open the **Segment Association** dialog box.
5. From the **Associate segment** list, select a segment with which you want to associate the current terminator.

TIP If the required segment is not available on the **Associate segment** list, select the **Include all segments** option button in the **Segment filter** group box.

6. Click **OK** to make the association.

See Also

Profibus Design Common Tasks (on page 213)

Terminator (Terminal Strip) - Segment Association

1. In the **Domain Explorer**, expand the hierarchy of a panel.
2. Right-click an appropriate fieldbus strip and then on the shortcut menu, click **Actions > Connection**.
3. In the **Connection** window, make sure that the required terminal strip is selected in the **Terminal strip** list.
4. Right-click the required terminator and on the shortcut menu, click **Associate Segment** to open the **Segment Association** dialog box.
5. From the **Associate segment** list, select a segment with which you want to associate the current terminator.

TIP If the required segment is not available on the **Associate segment** list, select the **Include all segments** option button in the **Segment filter** group box.

6. Click **OK** to make the association.

See Also

Profibus Design Common Tasks (on page 213)

Fieldbus Bricks

When designing your Foundation Fieldbus or Profibus system, you need to create a Fieldbus brick assembly. A fieldbus brick assembly is a composite object that consists of a wiring equipment item (fieldbus brick) and an apparatus group. An apparatus group constitutes an apparatus strip (that is, a terminal strip) with a number of apparatuses created according to a selected apparatus configuration.

When creating a fieldbus brick, you can define power supply requirements and current consumption. Adding an apparatus to a fieldbus brick, entails the definition of an apparatus configuration and the number of apparatuses that you need. Note that the power supply requirements and current consumption values are common to all the existing apparatuses belonging to the apparatus group that exists under a fieldbus brick. Therefore, changing an electric property values affects all the apparatuses that exist under that fieldbus brick. For more details about wiring equipment and apparatuses, see *Wiring Equipment* (on page 61).

When generating a segment validation report, the software takes into account the fieldbus brick electric properties and validates the total current of a segment.

We recommend that first you create your equipment in the **Reference Explorer** so that you have as many typical configurations as possible. Then, you can copy these typical configurations to the **Domain Explorer** and this way create numerous fieldbus bricks on the fly.

See Also

Profibus Design Common Tasks (on page 213)

SECTION 26

Managing Fieldbus Brick Assemblies and Apparatuses Common Tasks

The following tasks are used frequently when you manage fieldbus brick assemblies and apparatuses for your Foundation Fieldbus and Profibus systems.

Create a Fieldbus Brick Assembly

When designing your Foundation Fieldbus or Profibus system, you need to create a fieldbus brick assembly. A fieldbus brick assembly is a composite object that consists of a wiring equipment item (fieldbus brick) and an apparatus group. An apparatus group constitutes an apparatus strip (that is, a terminal strip) with a number of apparatuses created according to a selected apparatus configuration. When creating a fieldbus brick, you can define power supply requirements and current consumption. Note that if there are several apparatuses under a fieldbus brick, the power supply requirements and current consumption values are common to all the existing apparatuses belonging to that fieldbus brick. Therefore, changing an electric property values affects all the apparatuses that exist under that fieldbus brick.

We recommend that first you create your equipment in the **Reference Explorer** so that you have as many typical configurations as possible. Then, you can copy these typical configurations to the **Domain Explorer** and this way create numerous fieldbus bricks on the fly.

For more information, see *Create a Fieldbus Brick Assembly* (on page 195).

Configure a Fieldbus Apparatus

To create a fieldbus brick assembly, you must have a predefined apparatus group configuration. An apparatus group constitutes an apparatus strip (that is, a terminal strip) with a number of apparatuses created according to a selected apparatus configuration. When defining an apparatus configuration, you define the apparatus profile (name, description, manufacturer, and model), the number of apparatuses, the apparatus terminal configuration, and the position numbering. For more information, see *Configure a Fieldbus Apparatus* (on page 197).

Create a Fieldbus Brick Assembly

1. Press F8 to open the **Reference Explorer**.
2. Double-click the **Panels** folder, and select a panel.
3. Right-click the selected panel and then on the shortcut menu, click **New > Wiring Equipment**.
4. On the **New Wiring Equipment** dialog box, from the **Category** list, select **Fieldbus Brick**.
5. Under **Name**, type the name of the new fieldbus brick and click **OK**.
6. In the **Wiring Equipment Properties – Fieldbus Brick** dialog box, on the **General** tab, type the name of the new fieldbus brick.
7. Select the **Double width** check box if the fieldbus brick occupies a double width slot or position.
8. In the **Details** group box do the following as you require:
 - a. Type a description.
 - b. Select a type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.

- c. Enter a sequence if you need to define the card sequence.
9. Click the **Category Properties** tab to enter the power supply requirement and current consumption values for the reference fieldbus brick.

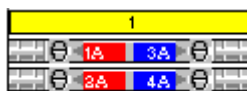
TIP Fieldbus brick is a wiring equipment category that is shipped with SmartPlant Instrumentation. You cannot delete or rename any of the category properties that have been shipped with the software. However, you can add user-defined properties which you can rename or delete as you wish. For details, see *Add User-Defined Wiring Equipment Category Properties* (on page 63). Note that the software uses the **Current consumption** and the **User-defined current consumption** values in segment validation reports.
10. Click the **Associated Symbols** tab to associate a symbol with this item. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **OK** to accept your settings and close the dialog box.
12. In the **Reference Explorer**, right-click the new fieldbus brick, then on the shortcut menu, click **New > Apparatus Group**.
13. On the **Apparatus Group** dialog box, select the desired configuration from the **Configuration name** list. Make sure that this configuration is compatible with fieldbus and that the **Fieldbus apparatus** check box is selected.

TIP If you do not have an appropriate apparatus configuration, create one. (For details, see *Configure a Fieldbus Apparatus* (on page 197)).
14. Enter the number of apparatuses that you require.
15. Click **Create**.

TIP SmartPlant Instrumentation automatically assigns sequential position numbers to each apparatus. You can edit them later if required. For details, see *Edit Apparatus Terminal Properties* (on page 76).
16. On the **Terminal Strip Properties** dialog box, define the new apparatus strip as required click **OK**. (For details, see *Create a Terminal Strip* (on page 85)).
17. Click **Close** in the **Apparatus** dialog box.

TIP The new apparatus appears in the **Reference Explorer** under the current **fieldbus brick**.
18. Drag the new fieldbus brick to a panel in the **Domain Explorer**.

TIP The software copies the fieldbus brick assembly that includes the fieldbus brick and the apparatus.
19. On the **Terminal Strip Properties** dialog box, rename the fieldbus brick, modify the other properties as you require, and then click **OK**.
20. Connect the apparatus to the appropriate cables. Right-click the apparatus strip you want to connect and then on the shortcut menu, click **Actions > Connection**. The **Connection** window opens displaying the apparatus terminals in yellow:



See Also

Profibus Design Common Tasks (on page 213)

Configure a Fieldbus Apparatus

To create a fieldbus brick assembly, you must have a predefined apparatus group configuration. An apparatus group constitutes an apparatus strip (that is, a terminal strip) with a number of apparatuses created according to a selected apparatus configuration. When defining an apparatus configuration, you define the apparatus profile (name, description, manufacturer, and model), the number of apparatuses, the apparatus terminal configuration, and the position numbering.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Do one of the following:
 - In the **Domain Explorer**, double-click the **Panels by Location** folder, and select a panel.
 - In the **Domain Explorer**, double-click the **Panels by Category** folder, and select a panel.
 - In the **Reference Explorer**, double-click the **Panels** folder, and select a panel.
3. Right-click a panel to which you want to add a fieldbus termination block and then on the shortcut menu, click **New > Apparatus Group**.
4. In the **Apparatus** dialog box, click **New**.
5. In the **Apparatus profile** group box, in the **Configuration name** field, type the name of the new apparatus configuration.
6. Select the **Fieldbus apparatus** check box.
7. Select the **Internal terminator** check box if you need to add an internal terminator on the new termination block.
8. In the **Description** field, type a short description for the new configuration.
9. From the **Manufacturer** list, select the appropriate termination block manufacturer. If the required value is not available, click  next to the list arrow to add or edit values for this list.
10. From the **Model** list, select the appropriate termination block model. If the required value is not available, click  next to the list arrow to add or edit values for this list.
11. Use the **Number of apparatuses** spinner or type the required number of termination blocks to be added. This way you can have a block of more than one apparatuses. This number will be displayed by default when adding a new termination block to a strip.
12. In the **Apparatus numbering** group box, type the apparatus name in the **Prefix** field if required.

TIP Apparatus names can be 15-character long. This name will appear in the **Domain Explorer** within the new strip (only the first four characters), in the **Connection** window, and in the appropriate reports.
13. Select the **Numbered** check box to number the apparatuses.
14. In the **Apparatus configuration** group box, from the **Orientation** list, select one of the following apparatus terminal patterns:
 - **Left / Right** — A termination block that has two sides: input and output (that is, left and right).
 - **Left** — A termination block that has wires on the left side only.
 - **Right** — A termination block that has wires on the right side only.
 - **Regular** — A termination block that has a conventional terminal that functions as a single unit.

15. Do the following to define the apparatus terminal name and color:
 - a. Click in the left and/or right terminal side text box and type the required terminal name, for example **1A**:
 - b. Select a color of the terminal side from the list, for example red:
16. Repeat steps 17 and 18 for the other terminal side if appropriate. Note that the line between the two terminal list arrows represents the internal relation between the input and output terminals.
17. Click **Add Terminal** to add another terminal to this apparatus configuration if needed and then repeat steps 14 through 16.
18. Click **Delete Terminal** if you need to delete a selected terminal from the current configuration.
19. Click **Save** and then **Close**. You can now add the required termination block as needed.

See Also

Profibus Design Common Tasks (on page 213)

SECTION 27

Making Connections and Associations for Fieldbus

This section deals with making connections and associations in your fieldbus system. Before you can associate a segment with a home-run cable or connect a home-run cable to a plug-and-socket box, you need to configure various connector types and assign connector pins to wires.

In This Section

Define or Edit a Connector Type	199
Define or Edit a Connector Type Pin Configuration	200
Delete a Connector Type.....	200
Create a Connector	200
Assign Connector Pins to Cable Wires.....	201
Associate a Pin or a Terminal with a Channel.....	201
Delete a Connector.....	202
Connect a Home-Run Cable to a Plug-and-Socket Box	202
Associate a Segment with a Home-Run Cable	203

Define or Edit a Connector Type

A connector type contains pin configuration and other properties for a connector model of a specific manufacturer. When you create a connector for a specific cable, selecting a connector type copies these properties for the cable connector.

1. On the Wiring module menu bar, click **Tables > Connector > Types**.
2. In the **Connector Types** dialog box, do one of the following:
 - To create a new connector type, click **New**.
 - To edit an existing connector type, click in the field that you want to edit.
3. Under **Connector Type**, type a unique name.
4. To specify the connector type for a specific manufacturer, model, and part number, do any of the following:
 - Under **Manufacturer**, select an appropriate connector manufacturer.
 - Under **Model**, select an appropriate model.
 - Under **Part Number**, type the part number you require.

NOTE You can now define a pin configuration for this connector type. For details, see *Define or Edit a Connector Type Pin Configuration* (on page 200).

See Also

Profibus Design Common Tasks (on page 213)

Define or Edit a Connector Type Pin Configuration

This topic explains how to define or edit a pin configuration for a connector type.

1. On the Wiring module menu bar, click **Tables > Connector > Types**.
2. In the **Connector Types** dialog box, select the connection type that you want to configure, and click **Configure**.
3. In the **Connector Type Configuration** dialog box, do one of the following for each pin that you want to configure:
 - To add a new pin, click **New**.
 - To delete a pin, select the row that you want to delete and click **Delete**.
 - To edit a pin field, click the field.
4. To change the displayed pin sequence, under **Sequence**, type the new unique number.
5. To set the pin name, under **Name**, type the unique name.
6. To set the pin polarity, under **Polarity**, select the value that you require.

See Also

Flow of Activities for Profibus Design (on page 211)

Delete a Connector Type

This topic shows how to delete a connector type.

1. On the Wiring module menu bar, click **Tables > Connector > Types**.
2. In the **Connector Types** dialog box, select the connector type that you want to delete.
3. Click **Delete**.

NOTE SmartPlant Instrumentation removes the item that you delete from the **Connector Types** dialog box from the **Connector type** list.

Create a Connector

Connectors are needed for fieldbus home-run cables, telecom cables, and other cables that require connectors. SmartPlant Instrumentation allows you to build your wiring system with any of the following connector implementations:

- Connector at one or both cable ends
 - Connector at end of a single cable set
 - Connector at end of wires from different cable sets
1. In the **Domain Explorer** or **Reference Explorer**, do one of the following:
 - To create a new cable, right-click the **Cables** folder and on the shortcut menu click **New > Cable**.
 - Right-click an existing cable, and on the shortcut menu click **Properties**.
 2. Modify cable properties as needed. For details, see *Create a Cable* (on page 28).
 3. Click **Connectors**.
 4. In the **Connector Assignments** dialog box, do one of the following:
 - To manage connector-wire associations for end 1 of the cable, click the **End 1** tab.
 - To manage connector-wire associations for end 2 of the cable, click the **End 2** tab.
 5. In the **Connector Assignments** dialog box, click **New**.
 6. In the **Connector Properties** dialog box that opens, under **Connector**, type the name of the new connector.

- From the list, select a connector type as configured.
 - To modify or create a connector type, click . For details, see *Define or Edit a Connector Type* (on page 199).
7. Under **Male/female** select **Male** or **Female** as needed.
 8. Click **OK** to return to the **Connector Assignments** dialog box.

Assign Connector Pins to Cable Wires

This procedure shows how to assign connector pins to cable wires.

1. In the **Domain Explorer** or **Reference Explorer**, do one of the following:
 - To create a new cable, right-click the **Cables** folder and on the shortcut menu click **New > Cable**.
 - Right-click an existing cable, and on the shortcut menu click **Properties**.
2. Modify cable properties as needed. For details, see *Create a Cable* (on page 28).
3. Click **Connectors**.
4. In the **Connector Assignments** dialog box, do one of the following:
 - To manage connector-wire associations for end 1 of the cable, click the **End 1** tab.
 - To manage connector-wire associations for end 2 of the cable, click the **End 2** tab.
5. Do one of the following:
 - To validate that pin polarities that you set for this cable end are consistent with wire polarities, select the **Polarity validation** check box.
 - To skip polarity validation, clear the **Polarity validation** check box.
6. For each connector pin, under **Wires per Pin**, select the number of wires that you want to assign to the pin.
7. Under **Cable Set**, select the **Cable Set** from which you are assigning a wire to the current pin. (Repeat this for each connector pin.)
8. Under **Wire**, select the wire that you assign to the pin. (Repeat this for each connector pin.)

NOTES

- The value that you enter under **Wires per Pin** determines the number of rows displayed for a given pin.
- The software does not allow you to edit pin-wire assignment properties of a connector that is connected (in the **Connection** window).

Associate a Pin or a Terminal with a Channel

SmartPlant Instrumentation enables you to associate terminals and pins with wiring equipment channels. Note that these terminals and the terminal strip must belong to the same parent wiring equipment. Also, if you want to associate pins, the pins must belong to a connector which is a child item of the parent wiring equipment.

After you drag a terminal or a pin to a wiring equipment channel, the software retains it in its original place from which you dragged it. However, after expanding the destination hierarchy, the software also shows the terminal or pin that you dragged. This means that the terminal or pin is physically located in its original place but it is associated with the item to which it was dragged.

IMPORTANT When dragging a terminal or pin to a wiring equipment channel, the software retains it in its original place from which you dragged it. However, after expanding the destination hierarchy, the software also shows the terminal or pin that you dragged. This means that the terminal or pin is physically located in its original place but it is associated with the item to which it was dragged.

1. Do one of the following:
 - Press F7 to open the **Domain Explorer**.
 - Press F8 to open the **Reference Explorer**.
2. Expand a hierarchy to display the terminal or pin that you want to associate.
3. Drag your selection to the required wiring equipment channel.

Delete a Connector

This topic shows how to delete a connector.

1. In the **Domain Explorer** or **Reference Explorer**, right-click a cable, and on the shortcut menu click **Properties**.
2. In the **Cable Properties** dialog box, click **Connectors**.
3. In the **Connector Assignments** dialog box, do one of the following:
 - To manage connector-wire associations for end 1 of the cable, click the **End 1** tab.
 - To manage connector-wire associations for end 2 of the cable, click the **End 2** tab.
4. In the **Connector Assignments** dialog box, select the connectors that you want to delete.
5. Click **Delete**.
6. Click **OK** to return to the **Connector Assignments** dialog box.

NOTE The software does not allow you to delete a connector that is already connected (in the **Connection** window).

Connect a Home-Run Cable to a Plug-and-Socket Box

After creating the required plug-and-socket boxes and home-run cables with connectors, you can start connecting the your home-run cables to plug-and socket boxes.

1. Press F7 to open the **Domain Explorer**.
2. Expand the **Cables** folder and select a home-run cable.
3. Right-click the selected cable and then on the shortcut menu, click **Actions > Connection**.
4. In the **Domain Explorer**, expand the hierarchy of the fieldbus cable that you want to connect to the current plug-and-socket box.
5. Drag the connectors to their ports in the current plug-and-socket box in the **Plug-and- Socket Box Connection** window.

TIP Before you start dragging a connector, make sure that the cable connector matches the port type on the plug-and-socket box.

6. In the **Cable Connection Options** dialog box, select the cable end that you want to connect and click **OK**.
7. Repeat the above to connect other cables as needed.

NOTE To disconnect a home-run cable from the current plug-and-socket box, highlight the cable that you want to disconnect and click .

See Also

Terminator (Terminal Strip) " Segment Association (see "*Terminator (Terminal Strip) - Segment Association*" on page 193)

Terminator (Plug-and-Socket Box) " Segment Association (see "*Terminator (Plug-and-Socket Box) - Segment Association*" on page 193)

Create a Home-Run Cable (on page 185)

Associate a Segment with a Home-Run Cable

After connecting a home-run-cable to a plug and socket box or a fieldbus brick, you can associate the home-run cable or its cable-set with a segment.

1. Press F7 to open the **Domain Explorer**.
2. Select and then right-click a fieldbus plug-and-socket box or a fieldbus brick.
3. On the shortcut menu, click **Actions > Connection**.
4. In the **Connection** or the **Plug- and-Socket Box Connection** window, click the required connected home-run cable or a home-run cable-set to select it. For example:



5. Right-click the highlighted home-run cable and on the shortcut menu, click **Associate Segment** to open **Segment Association** dialog box.
6. From the **Associate segment** list, select the segment with which you want to associate the current home-run cable.

TIP If the required segment is not available in the **Associate segment** list, select the **Include all segments** option button in the **Segment filter** group box.

7. Click **OK** to make the association.

NOTE The name of segment you just associated is propagated along the entire home-run cable.

See Also

Create a Home-Run Cable (on page 185)

Create a Foundation Fieldbus Segment (on page 182)

SECTION 28

Managing Fieldbus I/O Cards and I/O Assignment Common Tasks

The following tasks are used frequently when you manage fieldbus I/O cards and effect I/O assignment for your Foundation Fieldbus and Profibus systems.

Create a Fieldbus I/O Card

You need to create fieldbus I/O cards so that you can associate segments and effect I/O assignment.

You can create new I/O cards either in the **Domain Explorer** or the **Reference Explorer**. If you are creating an I/O card that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference I/O cards facilitate fast creation of plant I/O cards by copying an existing I/O card configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an I/O card that does not have a configuration that you intend to use frequently, we recommend that you create this card in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the I/O cards in the current <unit>.

For more information, see *Create a Fieldbus I/O Card* (on page 205).

Effect Fieldbus I/O Assignment

You can effect I/O assignment after you make the required connections. For more information, see *Effect Fieldbus I/O Assignment* (on page 206).

Create a Fieldbus I/O Card

1. In the **Domain Explorer** or **Reference Explorer**, right-click a DCS or PLC panel where you want to create a fieldbus I/O card.
2. On the shortcut menu, click **New > Wiring Equipment**.
3. In the **New Wiring Equipment** dialog box, under **Category**, select **I/O card**.
4. In the **Wiring Equipment Properties – I/O Card** dialog box, on the **General** tab, do one of the following to name the new I/O card:
 - Select the **Apply naming convention** check box selected if you want the software to name the new card automatically according to the naming conventions that are set for this type of equipment in the Administration module. For more information, see SmartPlant Instrumentation Administration Help, Domain Administration > Naming Conventions.
 - Clear the **Apply naming convention check box** and under **Name**, type the name of the new card. Note that if you do not clear the **Apply naming convention** check box, the software will ignore the name that you type under **Name**.
5. Select the **Double width** check box if the card occupies a double width slot or position.
6. In the **Details** group box, type a description as needed.
7. Select an I/O card type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.
8. Enter a sequence if you need to define the card sequence.
9. Click the **Control System** tab.

10. In the **Control system details** group box, under **I/O type**, select one of the following I/O types to determine the actual function of the new I/O card:
 - **Fieldbus**
 - **Profibus DP**
 - **Profibus PA**
11. Under **Module**, type the software address that the current card is assigned to.

TIP **Module**, **Controller/ Processor**, **Rack**, and **Slot** are field headers specific to panel manufacturers. Selecting a different manufacturer when editing the panel displays the headings used by that panel manufacturer. For details, see *Customize Manufacturer-Specific Field Headers*.
12. From the **Controller/Processor** list, select an I/O card controller. If the required controller is not available on the list, click  to define a new one.
13. Click the **Category Properties** tab.
14. Revise and modify category property values as you require. Click the value for each property and modify it as needed.

TIP A category property is a wiring equipment category property that you can define yourself. For details, see *Add User-Defined Wiring Equipment Category Properties* (on page 63).
15. Click the **Associated Symbols** tab to associate a symbol with this item. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
16. Click **OK** to accept your settings and close the dialog box.

See Also

Fieldbus Bricks (on page 194)

Profibus Design Common Tasks (on page 213)

Effect Fieldbus I/O Assignment

1. In the **Domain Explorer**, select the DCS and PLC cabinets for which you want to effect I/O assignment.
2. Right-click the panel that you selected and then on the shortcut menu, click **Actions > I/O Assignment**.
3. On the **I/O Assignment Type** dialog box, click **Segment I/O Assignment**.
4. In the **Segment I/O Assignment** window, under **Assignment details**, do the following for each I/O card that requires I/O assignment.
5. Under **I/O termination**, select the desired I/O card.
6. For each segment for which you want to effect I/O assignment, drag it from the **Segments** data window to the appropriate channel in the data window of the **I/O card details** pane.
7. To cancel fieldbus I/O assignment for a given segment, drag it from the data window of the **I/O card details** pane back to the **Segments** data window.

NOTES

- To rename a selected control system tag, click , and on the **Control System Tag Properties** dialog box, edit the tag properties.
- To view input/output information for the current I/O card, click .
- To generate an I/O Tag Assignment report, click .

Fieldbus Validation

SmartPlant Instrumentation validates that you design fieldbus segments according to basic connection rules and according to parameters that you set for a given segment:

On-the-Fly Connection Validations

Upon attempting to connect, the software verifies that:

- An instrument to be connected is not associated with another segment.
- A home-run cable to be connected is not already connected to another segment.
- You are not associating or wiring a non-intrinsically-safe instrument to an intrinsically safe segment. In this case, SmartPlant Instrumentation will warn, but allow.

Segment-Report Validations

You perform other validations by running a report from the **Fieldbus Segment Manager**. SmartPlant Instrumentation notifies you in the report of a failed validation in the following cases:

- There are fieldbus tags that are associated with a segment but not wired.
- There are fieldbus tags that have wiring, but are not yet connected to a segment.
- There is a connection discontinuity between the field and a DCS or an I/O terminal strip.
- The number of instruments in a segment exceeds the maximum specified for that segment.
- The number of devices on a given spur exceeds the segment-wide parameter profile.
- A spur length exceeds the maximum set for that spur.
- Total cable length exceeds the maximum you defined for the segment profile.
- There are less than two terminators in a segment.
- Actual voltage at instrument terminals is below the minimum operating voltage.
- Total segment current consumption exceeds the maximum specified value.
- Total capacitance exceeds segment maximum specified capacitance.

NOTES

- To define a cable type that supports fieldbus validation, make sure that in the **Cable Type Properties** dialog box, you enter the required values for capacitance, capacitance UOM (unit of measure), cable resistance, cable resistance UOM, length UOM, and maximum fieldbus length (for details, see *Create a Cable* (on page 28)).
- To define an instrument tag that supports fieldbus validation, make sure that on the **Tag Number** dialog box you enter valid information under **Fieldbus Tag Number Properties** *Create Foundation Fieldbus and Profibus Instruments* (on page 176)).
- SmartPlant Instrumentation validation supports basic fieldbus networks consisting of a single segment with no parallel branches and no repeaters.

SECTION 29

Fieldbus Validation Common Tasks

The following tasks are used frequently when you generate validation reports for your Foundation Fieldbus and Profibus systems.

Generate a Validation Report for One Segment

This procedure shows you how to generate a validation report for one fieldbus segment. Such a report allows you to verify that you created this segment according to general connection rules and the specific segment-wide parameter profile associated with each section. Note that SmartPlant Instrumentation will formulate this type of report in the unit of measure set on the **Cable Lengths** tab of the **Segment- Wide Parameter Profile Properties** dialog box. For more information, see *Generate a Validation Report for One Segment* (on page 209).

Generate a Validation Report for Multiple Segments

This procedure shows you how to generate a validation report for one of more fieldbus segments. Such a report allows you to verify that you created fieldbus segments according to general connection rules and the specific segment-wide parameter profile associated with each segment. For more information, see *Generate a Validation Report for Multiple Segments* (on page 210).

Generate a Segment Wiring Report

Use this procedure to generate an enhanced report for a fieldbus segment. Note that you must install the Enhanced Report Utility on your local machine to be able to generate the communication line report. For more information, see *Generate a Segment Wiring Report* (on page 210).

Generate a Validation Report for One Segment

1. In the **Domain Explorer**, expand the **Fieldbus Segments** folder.
2. Select a segment and then on the shortcut menu, click **Reports > Validation Report**.

NOTE SmartPlant Instrumentation formulates this type of report in the unit of measure set on the **Cable Lengths** tab of the **Segment-Wide Parameter Profile Properties** dialog box.

Generate a Validation Report for Multiple Segments

1. In the **Domain Explorer**, click the **Fieldbus Segments** folder.
2. In the **Items** pane of the **Domain Explorer**, select the segments that you require.
3. Right-click the selected segments and then on the shortcut menu, click **Reports > Validation Report**.

NOTE You can also generate a validation report from the **Fieldbus Segment Manager**.

Generate a Validation Report for Multiple Segments from the Fieldbus Tag Numbers Browser

1. On the Wiring module menu bar, click **Actions > Fieldbus Segment Manager**.
2. Do one of the following:
 - Click **Actions > Batch Validation**.
 - Click .
3. On the **Validation Reports** dialog box, under **Segment list**, select the segments that you require or click **Select all** to include all segments in the validation report.
4. Under **Unit of measure for the report**, select **Feet** or **Meters**.
5. Click **Print**.

NOTE You can also generate a validation report from the **Fieldbus Segments** folder.

Generate a Segment Wiring Report

1. In the **Domain Explorer**, expand the **Fieldbus Segments** folder.
2. Right-click a segment, and then on the shortcut menu, click **Reports > Enhanced Segment Wiring**.
3. At the print preview prompt, do one of the following:
 - Click **Yes** to open the print preview of the generated report. Choose this option if you want to modify or annotate the report.
 - Click **No** to start printing the report without displaying its print preview.

NOTES

- You must install the Enhanced Report Utility on your local machine to be able to generate the communication line report. For details, see Enhanced Report Utility Installation Guide.
- In the **General** tab folder of the **Preferences** dialog box, you can set SmartPlant Instrumentation to display a print preview always, never or with your approval.
- If you select not to preview a report, and you are using Acrobat Distiller as your default printer, make sure that Distiller is configured to enable report generation without prompting for the output file location. To do this, open the Distiller **Preferences** dialog box and under **Output Options**, clear the check boxes **Ask for PDF file destination** and **Ask to replace existing PDF file**.

Profibus Design

SmartPlant Instrumentation supports Profibus technology which is based on a two-level bus hierarchy, DP and PA Profibus segments. Each DP bus can connect multiple addressable link hardware devices that link various types of input and output devices. Every link device has a unique node number. The DP link devices that you can connect to a DP bus can belong to one of the following types:

- Motor drive (connecting a VFD to a DP bus).
- PLC or DCS panel that connects a conventional I/O system of analog and discrete field devices to a DP bus.
- DP/PA type that connects a lower PA (Process Automation) bus along with its PA field devices to the DP bus (similar to Foundation Fieldbus segments).

Profibus PA

Profibus PA segments and Profibus instruments have the same characteristics as Foundation Fieldbus segments and Foundation Fieldbus instruments.

Flow of Activities for Profibus Design

The following is a suggested flow of activities for Profibus design in SmartPlant Instrumentation. Note that you can carry out some of the procedures in a different order, depending on your preferences or requirements.

1. Set the basic definitions as follows:
 - a. Define Profibus instrument type profiles - Prior to creating new Profibus tag numbers, you need to customize wiring reference items and then define appropriate instrument type profiles in the Instrument Index module so that new tag numbers acquire Profibus properties. This results in automatic assignment of the required properties to the new tag numbers. For details, see *Define Foundation Fieldbus and Profibus Instrument Type Profiles* (on page 176).
 - b. Add a new browser view - In the Browser module, create a browser view with the required fields for the **Fieldbus Segment Manager**. This facilitates the assignment of tag numbers to segments. You can create multiple Browser views with different Style, Sort, and Filter functions. For example, by using the filter function you may create independent views for each segment, so that only tags of that segment will be displayed in its corresponding view. For more information, see *Add a New View*.
2. Create the required DP segments. For details, see *Create a Profibus DP Segment* (on page 214).
3. Create your PLC and/or DCS panels that are compatible with Profibus. For details, see *Create a DP-Compatible PLC or DCS Panel* (on page 215).
4. Associate the PLC and DCS panels with the DP segments. For details, see *Associate a PLC or DCS Panel with a Profibus DP Segment* (on page 215).
5. Create the required Profibus DP multi-input instruments. Create these multi-input instruments with their connected I/O's. For example, create a motor drive and insert it in an MCC cabinet. For details, see *Create a Multi-Input Instrument with Terminal Connections for a Fieldbus Segment* (on page 225) and *Create a Multi-Input Instrument with Plug-and-Socket Connections for a Fieldbus Segment* (on page 223).
6. Associate the DP motor drives with the DP segments.

- For more information, see *Associate Profibus DP Instruments with a DP Segment* (on page 214).
 - Effect the DP Segment I/O Assignment
 - Create the required PA segments. For more information, see *Create a Profibus PA Segment* (on page 214).
7. Create the Profibus PA instruments. You create these tags based on the instrument types that you defined, and edit the tag number properties as required. For more information, see *Create Foundation Fieldbus and Profibus Instruments* (on page 176).
 8. Associate the PA instruments with the PA segments. For details, see *Associate Profibus PA Instruments with a PA Segment* (on page 214).
 9. Generate a Profibus layout report. For details, see *Generate an Enhanced Profibus Layout Report* (on page 216).

SECTION 30

Profibus Design Common Tasks

The following is a suggested flow of activities for Profibus design in SmartPlant Instrumentation. Note that you can carry out some of the procedures in a different order, depending on your preferences or requirements.

Create a Profibus DP Segment

You need to create a DP bus prior to creating PA segments or any instruments that are associated with Profibus. This procedure explains how to create a DP bus.. For more information, see *Create a Profibus DP Segment* (on page 214).

Create a Profibus PA Segment

This procedure explains how to create a Profibus PA segment. Note that you can create a Profibus PA segment only if you already have an existing DP bus. For more information, see *Create a Profibus PA Segment* (on page 214).

Associate Profibus PA Instruments with a PA Segment

This procedure outlines the steps required to associate PA instruments with a PA segment. Note that an instrument can be associated with one segment only. For more information, see *Associate Profibus PA Instruments with a PA Segment* (on page 214).

Associate Profibus DP Instruments with a DP Segment

Use this procedure to associate a Profibus DP device or an instrument (for example, a motor drive) with a DP segment. For more information, see *Associate Profibus DP Instruments with a DP Segment* (on page 214).

Create a DP-Compatible PLC or DCS Panel

You need DP-compatible panels for your Profibus system. For more information, see *Create a DP-Compatible PLC or DCS Panel* (on page 215).

Associate a PLC or DCS Panel with a Profibus DP Segment

This procedure explains how to associate a PLC or a DCS panel with a DP segment. For more information, see *Associate a PLC or DCS Panel with a Profibus DP Segment* (on page 215).

Generate an Enhanced Profibus Layout Report

This report allows you to view a schematic drawing of a Profibus segment. The report includes one page per segment and shows all the associated devices and their node numbers. For more information, see *Generate an Enhanced Profibus Layout Report* (on page 216).

Create a Profibus DP Segment

1. In the **Domain Explorer**, right-click the **Fieldbus Segments** folder.
2. On the shortcut menu, click **New > Profibus DP Segment**.
3. In the **Profibus DP Segment Properties** dialog box, under **Segment**, type a unique name for the new segment.
4. Click **OK** to create the new segment.

Create a Profibus PA Segment

1. In the **Domain Explorer**, expand the **Fieldbus Segments** folder.
2. Right-click a DP segment and then on the shortcut menu, click **New > Profibus PA Segment**.
3. In the **Profibus PA Segment Properties** dialog box, type the segment name.
4. Select a segment-wide parameter profile if needed. You can click  to define a new segment-wide parameter profile if you need one. For details, see *Define Segment-Wide Parameters*.
5. Select the **Intrinsically safe** check box if you want to define the new segment as intrinsically safe.
6. Click **OK** to create the new segment.

Associate Profibus PA Instruments with a PA Segment

1. On the Wiring module menu bar, click **Actions > Fieldbus Segment Manager**.
2. In the **Fieldbus Segment Manager**, select one or more PA instruments.
3. Drag the instruments you selected to a PA segment in the **Fieldbus Segments** folder of the **Domain Explorer**.

NOTE To dissociate a PLC or DCS panel from a DP segment, in the **Domain Explorer**, right-click a panel in the **Fieldbus Segment** folder, then on the shortcut menu, click **Actions > Dissociate Item from Segment**.

Associate Profibus DP Instruments with a DP Segment

1. On the Wiring module menu bar, click **Actions > Fieldbus Segment Manager**.
2. Select the desired instruments in the browser.
3. Do one of the following:
 - On the browser toolbar, click  and then select an unassociated instrument in the **DP Instruments** pop-up window and drag it to a DP segment in the **Domain Explorer**.
 - In the **Fieldbus Segment Manager**, select an unassociated instrument and drag it to a DP segment in the **Domain Explorer**.

NOTE To dissociate an instrument from a DP segment, in the **Domain Explorer**, right-click an instrument in the **Fieldbus Segments** folder, and then on the shortcut menu, click **Actions > Dissociate Item from Segment**.

Create a DP-Compatible PLC or DCS Panel

You can connect several different types of devices to a DP segment. As DP segments are Ethernet busses, you can connect PLC or DCS panels to a DP segment via a special I/O card.

You create a PLC or DCS panel compatible with a DP segment exactly the same way you create any other PLC or DCS panel. While creating such a panel, make sure that you select the **Enable DP Profibus** check box so that you can associate this panel with a DP Profibus segment.

For details, see the following:

- *Create a Distributed Control System (DCS) Panel* (see "Create a DCS (Distributed Control System) Panel" on page 42)
- *Create a Programmable Logic Controller (PLC) Panel* (see "Create a PLC (Programmable Logic Controller) Panel" on page 43)

Associate a PLC or DCS Panel with a Profibus DP Segment

1. Start the Wiring module.
2. Do one of the following to open the **Fieldbus Segment Manager**.
 - Click **Actions > Fieldbus Segment Manager**.
 - On the module toolbar, click .
3. In the **Fieldbus Segment Manager**, do one of the following to open the **Panels** pop-up window:
 - On the main menu, click **View > Panels**.
 - On the **Fieldbus Segment Manager** toolbar, click .

TIP The **Panels** pop-up window displays all the PLC panels that have been enabled for use with DP Profibus and that have not been associated with a DP segment. If the pop-up window is empty, create a new PLC cabinet and make sure that you select the **Enable DP segment** check box on its properties dialog box.

4. In the **Domain Explorer**, expand the **Fieldbus Segments** hierarchy to display the available DP segments.

TIP To create a new DP segment, right-click the **Fieldbus Segments** folder in the **Domain Explorer** and then click **New**.

5. In the **Panels** pop-up window, select an appropriate PLC panel and drag it to the required DP segment in the **Fieldbus Segments** folder of the **Domain Explorer**.

NOTE To dissociate a PLC or DCS panel from a DP segment, in the **Domain Explorer**, right-click a panel in the **Fieldbus Segment** folder, then on the shortcut menu, click **Actions > Dissociate Item from Segment**.

Generate an Enhanced Profibus Layout Report

1. In the **Domain Explorer**, expand the **Fieldbus Segments** folder.
2. Right-click a Profibus segment.
3. On the shortcut menu, click **Reports > Enhanced Profibus Layout**.

SECTION 31

HART Instruments

HART (Highway Addressable Remote Transducer) technology is one of the first fieldbus digital communication techniques that were implemented in instrumentation. HART is a widely acceptable and recognized standard for digitally enhanced 4-20 milliamp smart instrumentation communication. This technology extends the 4-20 mA standard analog transmission with a superimposed digital signal that contains process and instrument data.

The advantage of HART instruments is that they can be connected to standard 4-20 milliamperes systems and, at the same time; make use of digital communication to collect additional data.

Flow of Activities for HART Instrumentation

The following is a suggested flow of activities for HART instrumentation in SmartPlant Instrumentation. Note that you can carry out some of the procedures in a different order, depending on your preferences or requirements.

1. Define the appropriate instrument types for HART instruments. When defining the profile for HART instrument types you must select HART AI or HART AO as the I/O type. For more information, see *Define Instrument Type Profiles for HART Instruments* (on page 218).
2. Create your HART instruments. HART instruments can be HART transmitters, HART IP converters, and so on. For details, see *Create a HART Instrument* (on page 218).
3. Create a virtual digital tag. Each HART instrument has a different set of digital signals. The digital signals depend on the I/O type (AI or AO) and on the instrument manufacturer who supplies the HART instruments with a specified variety of possible digital signals. For more information, see *Create a Virtual Digital Tag* (on page 220).
4. Create I/O cards compatible with HART instruments. For details, see *Create an I/O Card Compatible with HART Instruments* (on page 220).
5. Connect the HART instruments to the appropriate HART I/O cards. You make this connection by right-clicking an I/O card in the **Domain Explorer** and then clicking **Actions >Connection**.
6. Effect I/O assignment for HART instruments. For details, see *Effect I/O Assignment for HART Instruments* (on page 220).

Define Instrument Type Profiles for HART Instruments

1. On the Instrument Index module, click **Tables > Instrument Types**.
2. In the **Instrument Types** dialog box, from the **Process function** list, select an appropriate process function.
3. Do one of the following:
 - Click **New** to create a new instrument type. For details, see *Define an Instrument Type*.
 - In the data window, select an existing instrument type.
4. Click **Profile**.
5. In the **Instrument Type Profile** dialog box, on the **General** tab, under **Specification**, select an appropriate fieldbus specification.
6. Select **Include I/O type**, and then from the list, select one of the following:
 - **HART AI**
 - **HART AO**
7. On the **Wiring and Control System** tab, select the appropriate reference device panel and reference cable. For details, see *Define a Wiring and Controls System Instrument Type Profile*.
8. Click **OK** in the **Instrument Type Profile** dialog box.

Create a HART Instrument

Press F7 to open in the **Domain Explorer**.

1. Expand the plant hierarchy to display the **Instruments** and **Loops** folders.
2. Do one of the following:
 - To create an instrument not associated with a loop number, right-click the **Instruments** folder, and then click **New > Instrument**.
 - To create an instrument that is associated with a specific loop number, expand the **Loops** folder, right-click a loop, and then click **New > Instrument**.
3. On the **New Tag Number** dialog box, do the following:
 - a. From the **Tag class** list, select **HART**.
 - b. Type the name of the new tag number.
 - c. Click **OK**.
4. On the **Select Instrument Type** dialog box, select the instrument type you want to associate with the current hart instrument, and then, click **OK**.
5. Click **OK**.

TIPS

- Remember that you need to define appropriate instrument type profiles for your HART instruments prior to creating them. These instrument type profiles must include an I/O type which is either HART AI or HART AO. For details, see *Define Instrument Type Profiles for HART Instruments* (on page 218).
- If the tag number does not correspond to an existing loop name, SmartPlant Instrumentation prompts you to enter a loop name based on the tag number you have entered. If the loop identifier already matches an existing loop, the software will automatically associate the new tag number with the loop.

- If the loop convention includes the loop function parameter, the prompt will always appear. In this case, you have to complete the loop number.
If a profile exists for the selected instrument type, any new tags for that instrument type will be created with the selected reference items.
 - For DeltaV compatibility, you must select an instrument type that you associated with a DeltaV device type during the downloading process. To display the **Associate Foundation Fieldbus Device Types with Instrument Type** supporting table of associations that you built among DeltaV device types, instrument manufacturers, and SmartPlant Instrumentation instrument types, see Associate Device Types for DCS Vendors.
6. On the **Loop Name** dialog box, do one of the following:
 - Type the loop number that the new tag is associated with.
 - Accept the displayed loop number.
 - Click **Cancel** to create the tag number without a loop association. Note that if a loop with the same name exists, the tag number is automatically associated with it, without creating a new loop.
 7. Click **OK** to create the loop number.
 8. On the **Loop Number Properties** dialog box, accept the loop number properties or modify them as you require and then click **OK**.
 9. On the **Tag Number Properties** dialog box, on the **General** tab, enter the tag number attributes that you require.
TIP For DeltaV compatibility, make sure that the value that you select from the **Manufacturer** list is among the acceptable values for the instrument type that you selected for the current tag. For details, see Crucial Fields for the DeltaV Interface.
 10. To enter power supply properties, see *Enter Power Supply Data for Panels and Instrument Tags* (on page 45).
 11. Click the **HART** tab to define the instrument parameters and create virtual digital tags.
 12. Select a signal type and linearity type values if needed.
 13. From the **Analog signal parameter** list, select a parameter that is appropriate for the analog signal of the current HART instrument.
TIP This selection is available only if the I/O type of the current instrument is **HART AI**.
 14. Enter the required DC consumption, capacitance, minimum transmit level, and operating voltage.
 15. Create the required virtual digital tags. For details, see *Create a Virtual Digital Tag* (on page 220).

Create a Virtual Digital Tag

1. In the **Tag Number Properties** dialog box, click the **HART** tab.
2. From the **Analog signal parameter** list, select an appropriate signal parameter.
3. In the **Virtual tags** group box, click **New**.
4. Click **Yes** in the prompt to save the changes to the tag properties.
5. In the **New Virtual HART Tag Number** dialog box, select a signal parameter, enter the tag number name.
6. Click **OK**.

NOTE The signal parameter depends on whether the I/O type of the current HART instrument is **HART AI** or **HART AO**.

Create an I/O Card Compatible with HART Instruments

To make an I/O card compatible with a HART instrument, you must select one of the following I/O types when defining the control system properties of a new I/O card:

- HART AI
- HART AO
- AI
- AO
- MIXED

For details on how to create an I/O card, see *Create an I/O Card* (on page 68).

Effect I/O Assignment for HART Instruments

You effect I/O assignment for your HART instruments like with any other instruments that require I/O assignment. In the **Domain Explorer**, right-click the required I/O card and then click **Actions > I/O Assignment**. For details, see *I/O Assignment* (on page 105).

Instruments	Cards			
	AI	AO	HART AI	HART AO
AI	Yes	No	Yes (see note 1)	No
AO	No	Yes	No	Yes (see note 1)
HART AI	Yes (see note 2)	No	Yes	No
HART AO	No	Yes (see note 2)	No	Yes
Mixed	Yes	Yes	Yes (see note 3)	Yes (see note 3)

NOTES

- When assigning "non-HART" analog instruments to HART-compatible I/O cards, the software assigns only the analog signals. (DI or DO are not allowed anyway).
- You can assign HART instruments to conventional I/O cards (AI to HART AI, AO to HART AO), but the software assigns only the analog signals and not the virtual digital tags.

- You can assign HART instruments to I/O cards with MIXED I/O type, but the software assigns only the analog signals and not the virtual digital tags.

Conventional and Fieldbus Multi-Input Devices

Many modern fieldbus or conventional digital instruments support more than one input, for example, temperature transmitters and Bentley Nevada vibration monitors. SmartPlant Instrumentation supports various configurations for instrument tags with multi-input device panels. Multi-input devices can recognize input signals from distant device panels.

Managing Multi-Input Devices

The following tasks are used frequently when you create and manage the fieldbus and conventional multi-input devices in your plant.

You can create new multi-input devices either in the **Domain Explorer** or the **Reference Explorer**. If you are creating a multi-input device that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference multi-input devices facilitate fast creation of plant multi-input devices that are based on the configuration of an item in the **Reference Explorer**. However, if you want to create a multi-input device that does not have a configuration that you intend to use frequently, we recommend that you create this multi-input device in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

Create a Multi-Input Device Panel with Terminal Connections

This procedure explains how to create a multi-input device panel that has terminal connections. For more information, see *Create a Multi-Input Device Panel with Terminal Connections* (on page 223).

Create a Plug-and-Socket Multi-Input Device Panel

This procedure shows you how to create a plug-and-socket multi-input device panel. For more information, see *Create a Plug-and-Socket Multi-Input Device Panel* (on page 222).

Create a Multi-Input Instrument with Plug- and-Socket Connections for a Fieldbus Segment

You use this procedure to create a multi-input instrument with plug-and-socket connections for a fieldbus segment and then assign the signals to fieldbus I/O card channels. For more information, see *Create a Multi-Input Instrument with Plug- and-Socket Connections for a Fieldbus Segment* (see "Create a Multi-Input Instrument with Plug-and-Socket Connections for a Fieldbus Segment" on page 223).

Create a Multi-Input Instrument with Terminal Connections for a Fieldbus Segment

This procedure explains how to create a multi-input instrument with terminal connections for a fieldbus segment. Also, you can assign the signals to fieldbus I/O card channels. For more information, see *Create a Multi-Input Instrument with Terminal Connections for a Fieldbus Segment* (on page 225).

Create a Multi-Input Instrument with Plug- and-Socket Connections for a Serial Loop Interface

This procedure explains how to create a multi-input instrument with plug-and-socket connections for a serial interface loop. Also, you can assign the signals to fieldbus I/O card channels and generate an enhanced report for the wiring loop. For more information, see *Create a Multi-Input Instrument with Plug-and-Socket Connections for a Serial Loop Interface* (on page 226).

Create a Multi-Input Instrument with Terminal Connections for a Serial Loop Interface

This procedure explains how to create a multi-input instrument with terminal connections for a serial interface loop. Also, you can assign the signals to fieldbus I/O card channels and generate an enhanced report for the wiring loop. For more information, see *Create a Multi-Input Instrument with Terminal Connections for a Serial Loop Interface* (on page 227).

Create a Plug-and-Socket Multi-Input Device Panel

1. In the **Domain Explorer** or **Reference Explorer**, do one of the following:
 - In the **Domain Explorer**, expand the **Panels by Category** folder.
 - In the **Reference Explorer**, expand the **Panels** folder.
2. Right-click the **Device Panels** folder and then on the shortcut menu, click **New > Multi-Input (Plug-and-Socket)**.
3. In the **Plug-and-Socket Box Wizard - Multi-Input Device**, read the instructions carefully and then click **Next** to open the **Define the New Panel** page.
4. Under **Panel name**, type a unique device panel name.
5. Define the new panel by entering the panel type, manufacturer, model, area class, location, mounting, dimensions, and backplane as needed.
6. To define the device panel as intrinsically safe, select **Intrinsic safety**.
7. Click **Next**, and on the **Define the Plug-and-Socket Layout** page, enter the number of ports that you need on the left and on the right sides of the panel.
8. Click **Next**, and on the **Define the Ports** page, for each port that you created, type the name and select **In** or **Out**.
TIP You must define one and only one out port.
9. Click **Next**, and on the **Define the Plug-and-Socket Box Connectors** page, for each connector, do the following:
 - a. From the **Connector type** list, select the required connector, or click  to open the **Connectors** dialog box.
 - b. From the **Male/female** list, define the connector as **Male** or **Female**.
 - c. If these settings are constant for all of the connectors in the panel, select **Apply to all** after you set the above.
10. Click **Next** and then click **Finish**.

NOTE You can delete the terminals that were created after completing the wizard only when you delete the multi-input device panel.

Create a Multi-Input Device Panel with Terminal Connections

1. In the **Domain Explorer** or **Reference Explorer**, do one of the following:
 - In the **Domain Explorer**, expand the **Panels by Category** folder.
 - In the **Reference Explorer**, expand the **Panels** folder.
2. Right-click the **Device Panels** folder and then on the shortcut menu, click **New > Multi-Input (Conventional)**.
3. In the **Conventional Box Wizard - Multi-Input Device**, read the instructions carefully and then click **Next** to open the **Define the New Panel** page.
4. Under **Panel name**, type a unique device panel name.
5. Define the new panel by entering the panel type, manufacturer, model, area class, location, mounting, dimensions, and backplane as needed.
6. To define the device panel as intrinsically safe, select **Intrinsic safety**.
7. Click **Next**, and on the **Define the Multi Input Layout** page, enter the number of inputs and outputs that you need on the left and on the right sides of the panel.
8. Click **Next**, and on the **Define the Multi Input** page, for each multi input that you created, type the name and select **In** or **Out**.
TIP Only one of the input/outputs can be defined as an output.
9. Click **Next**, and on the **Define the Input / Output Configuration** page, for each connector, do the following:
 - a. From the **Input/Output** list, select the required input or output.
 - b. From the **Configuration** list, select an appropriate configuration.
 - c. Under **First terminal**, enter the first terminal as you require.
 - d. If these settings are constant for all of the inputs and outputs in the panel, select **Apply to all** after you set the above.
10. Click **Next** and then click **Finish**.

NOTE You can delete the terminals that were created after completing the wizard only when you delete the multi-input device panel.

Create a Multi-Input Instrument with Plug-and-Socket Connections for a Fieldbus Segment

1. In the **Reference Explorer**, create reference panels that feed into the multi-input device panel.
2. In the **Reference Explorer**, create a reference multi-input device panel with plug- and-socket connections.
3. In the **Reference Explorer**, create reference spur cables.

TIPS

- Use the same connector type required for the device panels.
- Set the male/female property for coupling with the device panels.
- On the **Cable Properties** dialog box, under **Cable class**, select **Conventional**.

4. In the Instrument Index module, create instrument type profiles for multi-input plug-and-socket socket fieldbus instruments and instruments that feed into the multi- input instrument.

TIPS

- For each instrument type, on the **Wiring and Control System** tab, select the reference device panel and reference cable that you created for each type.
 - For the instrument type upon which you base the multi-input device panel, make sure that you select the **Fieldbus** I/O type on the **Instrument Type Profile** dialog box and that you associate enough function blocks for the inputs. For details, see *Define Foundation Fieldbus and Profibus Instrument Type Profiles* (on page 176).
5. In the Instrument Index module, create the following, based on the instrument types that you defined:
 - Create the instrument tags that feed into the multi-input instrument tag.
 - Create the multi-input instrument tag. Make sure that on the **New Tag Number** dialog box, you select **Fieldbus** from the **Tag class** list.
 6. In the **Domain Explorer**, right click the multi-input device panel that was created, and on the shortcut menu, click **Actions > Connection**.
 7. In the **Plug-and-Socket Connection** window, do the following to connect the input device cables to the multi-input device panel.
 - a. In the **Domain Explorer**, expand the device cables that feed into the multi- input device panel.
 - b. Drag the appropriate cable connectors to the in-ports of the multi-input device.
 8. Do the following to route the multiplexed signal through a plug-and-socket junction box:
 - a. In the **Domain Explorer**, right click the junction box, and on the shortcut menu, click **Actions > Connection**.
 - b. In the **Domain Explorer**, expand the multi-input device cable.
 - c. Drag the appropriate cable connector to the desired in-port of the junction box in the **Plug-and-Socket Connection** window.
 9. Do the following to associate the multi-input tag signal with the available function blocks in the target fieldbus segment:
 - a. In the **Wiring Module** window, click **Actions > Fieldbus Segment Manager**.
 - b. Drag the multi-input device tag from the **Instruments** pane to the target segment in the **Fieldbus Segments** folder of the **Domain Explorer**.
 - c. In the **Fieldbus Segments** pane, right click the multi-input device tag, and on the shortcut menu, click **Associate Multi-Input Device Tag**.
 - d. Drag each tag from the **Connected tags available for association** pane to the **Function Block - Tag Association** pane.
 - e. Click **Close**.
 10. Do the following to assign the demultiplex signals to the segment I/O card:
 - a. In the **Domain Explorer**, right-click the desired DCS, then on the shortcut menu click **Actions > I/O Assignment**.
 - b. If the **I/O Assignment** dialog box opens, select **Segment I/O assignment**, and click **OK**.
 - c. Select the desired segment in the **Segment list** pane of the **Segment I/O Assignment** window.
 - d. In the **I/O card details** pane, select the channel to which you want to assign the demultiplex signals.
 - e. Click **Actions > Assign to a Channel**.

Create a Multi-Input Instrument with Terminal Connections for a Fieldbus Segment

1. In the **Reference Explorer**, create reference panels that feed into the multi-input device panel.
2. In the **Reference Explorer**, create a reference multi-input device panel with terminal connections.
3. In the **Reference Explorer**, create reference spur cables.

TIP In the **Cable Properties** dialog box, under **Cable class**, select **Conventional**.
4. In the Instrument Index module, create instrument type profiles for multi-input plug-and-socket socket fieldbus instruments and instruments that feed into the multi-input instrument.

TIPS

 - For each instrument type, on the **Wiring and Control System** tab, select the reference device panel and reference cable that you created for each type.
 - For the instrument type upon which you base the multi-input device panel, make sure that you select the **Fieldbus I/O** type in the **Instrument Type Profile** dialog box and that you associate enough function blocks for the inputs. For details, see *Define Foundation Fieldbus and Profibus Instrument Type Profiles* (on page 176).
5. In the Instrument Index module, create the following, based on the instrument types that you defined:
 - Create the instrument tags that feed into the multi-input instrument tag.
 - Create the multi-input instrument tag. Make sure that in the **New Tag Number** dialog box, you select **Fieldbus** from the **Tag class** list.
6. In the **Domain Explorer**, right click the multi-input device panel that was created, and on the shortcut menu, click **Actions > Connection**.
7. In the **Plug-and-Socket Connection** window, connect the input device cables to the multi-input device panel.
8. Do the following to route the multiplexed signal through a junction box:
 - a. In the **Domain Explorer**, right click the junction box, and on the shortcut menu, click **Actions > Connection**.
 - b. In the **Connection** window, connect the multi-input device cable to the junction box.
9. Do the following to associate the multi-input tag signal with the available function blocks in the target fieldbus segment:
 - a. In the Wiring Module window, click **Actions > Fieldbus Tag Numbers Browser**.
 - b. Drag the multi-input device tag from the **Instruments** pane to the target segment in the **Fieldbus Segments** folder of the **Domain Explorer**.
 - c. In the **Fieldbus Segments** pane, right click the multi-input device tag, and on the shortcut menu, click **Associate Multi-Input Device Tag**.
 - d. Drag each tag from the **Connected tags available for association** pane to the **Function Block - Tag Association** pane.
 - e. Click **Close**.
10. Do the following to assign the demultiplex signals to the segment I/O card:
 - a. In the **Domain Explorer**, right-click the desired DCS, then on the shortcut menu click **Actions > I/O Assignment**.

- b. If the **I/O Assignment** dialog box opens, select **Segment I/O assignment**, and click **OK**.
- c. In the **Segment I/O Assignment** window, in the **Segment list** pane, select the desired segment.
- d. In the **I/O card details** pane, select the channel to which you want to assign the demultiplex signals.
- e. Click **Actions > Assign to a Channel**.

Create a Multi-Input Instrument with Plug-and-Socket Connections for a Serial Loop Interface

1. In the **Reference Explorer**, create reference panels that feed into the multi-input device panel.
2. In the **Reference Explorer**, create a reference multi-input device panel with plug- and-socket connections.
3. In the **Reference Explorer**, create reference spur cables.

TIPS

- For both types, in the **Cable Properties** dialog box, under **Cable class**, select **Conventional**.
 - Use the same connector type required for the device panels.
 - Set the male/female property for coupling with the device panels.
 - For the instrument type that you will use with the multi-input device panel, create one connector only (for the device panel end). Leave the second end without a connector, for terminal connection to the DCS I/O card.
4. In the Instrument Index module, create instrument type profiles for instruments with plug-and-socket multi-input serial interface and instruments that feed into the multi- input instrument.

TIPS

- For each instrument type, on the **Wiring and Control System** tab, select the reference device panel and reference cable that you created for each type.
 - For the instrument type upon which you base the multi-input device panel, make sure that you select the **Serial Interface** I/O type in the **Instrument Type Profile** dialog box.
5. In the Instrument Index module, create the following, based on the instrument types that you defined:
 - Create the instrument tags that feed into the multi-input instrument tag.
 - Create the multi-input instrument tag. Make sure that in the **New Tag Number** dialog box, you select **Fieldbus** from the **Tag class** list.
 6. In the **Domain Explorer**, right click the multi-input device panel that was created, and on the shortcut menu, click **Actions > Connection**.
 7. In the **Plug-and-Socket Connection** window, do the following to connect the input device cables to the multi-input device panel.
 8. In the **Domain Explorer**, expand the device cables that feed into the multi- input device panel.
 9. Drag the appropriate cable connectors to the in-ports of the multi-input device.
 10. In the **Domain Explorer**, right- click the DCS panel to which you connect the output of the multi-input device panel, then on the shortcut menu, click **Actions > Connection**.

TIP Make sure that the I/O type of the I/O card to which you connect the multi-input device is **SI**.

11. In the **Connection** window, connect the output cable of the multi-input device panel to the DCS.
12. To view the I/O assignment of the signals, click **Actions > I/O Assignment**.

Create a Multi-Input Instrument with Terminal Connections for a Serial Loop Interface

1. In the **Reference Explorer**, create reference panels that feed into the multi-input device panel.
2. In the **Reference Explorer**, create a reference multi-input device panel with terminal connections.
3. In the **Reference Explorer**, create reference spur cables.

TIPS

- For both types, in the **Cable Properties** dialog box, under **Cable class**, select **Conventional**.
- Do not create connectors for this reference cable.

4. In the Instrument Index module, create instrument type profiles for multi-input instruments with terminal connections for a serial interface and instruments that feed into the multi-input instrument.

TIPS

- For each instrument type, on the **Wiring and Control System** tab, select the reference device panel and reference cable that you created for each type.
 - For the instrument type upon which you base the multi-input device panel, make sure that you select the **Serial Interface** I/O type in the **Instrument Type Profile** dialog box.
5. In the Instrument Index module, create the following, based on the instrument types that you defined:
 - Create the instrument tags that feed into the multi-input instrument.
 - Create the multi-input instrument tag. Make sure that in the **New Tag Number** dialog box, you select **Fieldbus** from the **Tag class** list.
 6. In the **Domain Explorer**, right click the multi-input device panel that was created, and on the shortcut menu, click **Actions > Connection**.
 7. In the **Plug-and-Socket Connection** window, do the following to connect the input device cables to the multi-input device panel.
 8. In the **Domain Explorer**, expand the device cables that feed into the multi-input device panel.
 9. Drag the appropriate cable connectors to the in-ports of the multi-input device.
 10. In the **Domain Explorer**, right-click the DCS panel to which you connect the output of the multi-input device panel, then on the shortcut menu, click **Actions > Connection**.

TIP Make sure that the I/O type of the I/O card to which you connect the multi-input device is **SI**.

11. In the **Connection** window, connect the output cable of the multi-input device panel to the DCS.
12. To view the I/O assignment of the signals, click **Actions > I/O Assignment**.

SECTION 32

Telecom Design

SmartPlant Instrumentation provides the ability to efficiently define and manage most commonly used telecommunication equipment.

Here are a few examples of telecom equipment that you can create and manage:

- Field equipment (speakers, intercoms, telephones, and so forth)
- Public announcement (PA) equipment and amplifiers
- Switchboards
- Network equipment (hubs, switches, routers, and so forth)
- Miscellaneous equipment (video and entertainment equipment)

The software provides the means to define all the connections and the internal equipment configuration. Furthermore, you can define and create the logical structures (for example, channels, port, and so forth) as well the physical structures, such as racks and cards.

You can generate appropriate reports, such as telecommunication field lists, communication line diagrams (telecom point-to-point diagrams), network class diagrams (network area maps), and so forth.

Flow of Activities for Telecom Design

The following is a suggested flow of activities that will help you design the telecommunication system in your <plant>.

The first step in your telecom design is to create the reference telecommunication equipment. Then, in the Instrument Index module, you define telecommunication device types (similar to instrument types for non-telecommunication devices). You can also define telecom line numbers, field equipment (that is, equipment classification), and signal levels. Now you can create your telecom tag numbers based on the telecom device type that you defined. At this stage, you can generate various Instrument Index reports as needed.

Once all the supporting table data is entered and all the telecom tags are created, you start creating your telecom panels and telecom equipment based on the reference equipment and reference panels you created previously. Make the required connections among the your telecom equipment. You can then generate telecom wiring reports as needed.

We recommend the following flow of activities:

1. In the **Reference Explorer**, create the following reference equipment:
 - a. Telecom cables with appropriate configurations
 - b. Telecom device panels
 - c. Telecom equipment panels with or without their internal equipment
 - d. Telecom equipment items such as amplifiers, switchboards, hubs, and so forth that serve as the internal equipment of telecom panels
2. Start the Instrument Index module and define the following:

- a. A telecom device panel type and device type profile defaults for telecom tags
 - b. Communication line numbers
 - c. Field equipment classification
 - d. Signal levels
 - e. Generate Instrument Index telecom reports if needed (telecom devices and various telecom table reports).
3. In the **Domain Explorer**, do the following:
 - a. Create the telecom tag numbers that you require.
 - b. Create plant equipment panels with the necessary hardware shelves or racks.
 - c. Create the telecom equipment items that you require by copying the reference items.
 - d. Define network classes.
 - e. Make all necessary connections.
 - f. Generate the Telecom reports in the Wiring module.

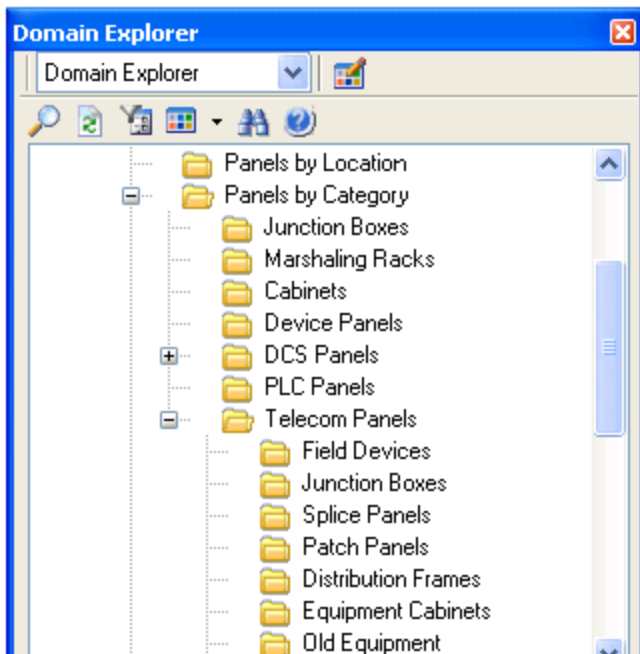
See Also

Working with Old Equipment (on page 232)

Arrangement of Telecom Items in the Domain Explorer

You use the **Domain Explorer** to access telecom panels and equipment. Typical telecom equipment, that is, telecom reference equipment, is organized in the **Reference Explorer**.

Telecom equipment is organized in several different folders in the **Domain Explorer**:



- **Telecom Panels** — contains telecom panels arranged according to their types:
- **Field Devices** — holds plug-and-socket boxes and conventional field device panels.
- **Junction Boxes**
- **Splice Panels**
- **Patch Panels**
- **Distribution Frames**

- **Equipment Cabinets** — holds PA cabinets, PABX cabinets, hub cabinets, intercom cabinets, and miscellaneous cabinets for video and entertainment systems)
- **Old Equipment** — contains telecom panels created prior to Version 7. Note that you cannot create additional panels in this folder, but you can add new child items, modify the properties of the existing items, and delete the old equipment items, as you require.
- **Panels by Manufacturer** — arranges telecom panels according to their manufacturers.
- **Telecom Equipment** — lists telecom panels and equipment classified according to their types. Note that the items in this folder are for viewing only.

In the **Reference Explorer**, reference telecom items reside under **Wiring Equipment**.

Supporting Tables for Telecom

The following table lists all the available Telecom supporting tables and the description of their functions. You can access these supporting tables by clicking  next to a list arrow or by clicking **Tables > Telecom** and then an appropriate command in the Instrument Index and Wiring modules.

Supporting Table	Description
Telecom Device Types	Allows you to create and manage telecom device types. You define the required telecom device type profile for your telecom tag numbers so that new tags that you create can acquire the necessary properties. SmartPlant Instrumentation provides a number of predefined telecom device types, such as AL (alarm), CAM (camera), and so forth. You can add new telecom device types and modify the shipped ones as desired. You can also delete a telecom device type that is not in use, but the software does not let you delete a telecom device type that you already used to create a tag number.
Telecom Line Numbers	Allows you to maintain the contents of the Telecom line list when creating a telecom field device in the Plug-and-Socket Box wizard and the Line number list on the Tag Number Properties dialog box.
Telecom Field Equipment	Allows you to maintain the contents of the Field equipment select list on the Tag Number Properties dialog box.
Telecom Signal Levels	Allows you to maintain the contents of the Signal level select list on the Tag Number Properties dialog box.
Panel Sub-Systems	Allows you to maintain the contents of the Sub-system select list on the Equipment Panel Properties dialog box.
Telephone Numbers	Allows you to define new telephone numbers that you can associate with existing switch channels in PABX cabinets.
Telephone Number Statuses	Allows you to manage the contents of the Telephone number status select list on the Telephone Number Properties dialog box.
Telephone Number Usages	Allows you to manage the contents of the Telephone number usages select list on the Telephone Number Properties dialog box.
Intercom Numbers	Allows you to define new intercom numbers that you can later associate with amplifiers.

Supporting Table	Description
PA Zones	Allows you to manage the PA zone definitions which are available for selection on the Category Properties tab of the Wiring Equipment Properties (Amplifier) dialog box.
Alarm Zones	Allows you to manage the alarm zone definitions which are available for selection on the Category Properties tab of the Wiring Equipment Properties (Amplifier) dialog box.
PABX Categories	Allows you to manage the contents of the PABX categories select list on the PABX Cabinet Properties dialog box.

NOTE For the various actions that you can perform with supporting tables, see Supporting Tables.

Working with Old Equipment

SmartPlant Instrumentation partially supports telecom panels created prior to Version 7. These panels and their child items are organized in the **Old Equipment** folder.

Note that you cannot create additional panels in this folder, but you can add new child items, modify the properties of the existing items, and delete the old equipment items, as you require.

See Also

Arrangement of Telecom Items in the Domain Explorer (on page 230)

Flow of Activities for Telecom Design (on page 229)

SECTION 33

Telecom Devices and Cables

This section describes the creation and management of telecom devices and cables.

In This Section

Set Telecom Device Type Profile Defaults	233
Create a Telecom Field Tag	234
Duplicate a Telecom Field Tag	234
Create a Telecom Conventional Field Device	235
Create a Telecom Plug-and-Socket Field Device	236
Create a Telecom Cable	236

Set Telecom Device Type Profile Defaults

You define a telecom device type profile for your telecom devices so that new telecom tag numbers that you create acquire the necessary properties. SmartPlant Instrumentation provides a number of predefined telecom device types, such as AL (alarm), CAM (camera), and so forth. You can add new telecom device types and modify the shipped ones as required. You can delete a telecom device type that is not in use. However, SmartPlant Instrumentation does not let you delete a telecom device type that you already used to create a tag number.

SmartPlant Instrumentation uses standard function identifier acronyms to identify telecom device types. If some acronyms are not unique and are used for more than one device type, the **Select Telecom Type** dialog box opens where you select the appropriate device type.

After defining a device type, you set device type profile defaults for new telecom tag numbers. As result, SmartPlant Instrumentation will create your new tag numbers with certain predefined properties based on the telecom device type you select for the new tag number. You set profiles for every telecom device type that you use.

1. On the **Instrument Index Module** window menu bar, click **Tables > Telecom > Telecom Device Types**.
2. On the **Telecom Device Types** dialog box, do one of the following:
 - Select an existing device type and click **Profile**.
 - Click **New** and define a device type that you need and then click **Profile**.
3. On the **Telecom Device Type Profile** dialog box, define the general default settings for new telecom devices.
4. Click the **Wiring** tab if you need to define wiring default settings.
5. On the **Wiring and Control System** tab, select the **Include wiring** check box.
6. To define new device tags as control-system-enabled, select the **Control system** check box. This enables the corresponding tag numbers for tag assignment.
7. To enable automated creation of coupled control system tags, select **Automatic CS tags**.
8. From the **Reference device panel** list, select an appropriate reference panel for this instrument type. These are device panels that were created in **Reference Explorer**.
9. If your reference device panel settings enable the **Conventional connections** group box, do one of the following for each connection that you need for the current instrument type:
 - Click **New**, to add a connection.

- Click **Properties**, to edit the properties of an existing convention, and on the **Conventional Connection Properties** dialog box, define the settings that you require, and click **OK**.
10. If your reference device panel settings enable the **Plug-and-socket connections** group box, do one of the following for each connection that you need for the current instrument type:
 - Click **New**, to add a connection.
 - Click **Properties**, to edit the properties of an existing convention, and on the **Plug-and-Socket Connection Properties** dialog box, define the settings that you require, and click **OK**.

Create a Telecom Field Tag

This procedure explains how to create new telecom tag numbers. Note that the new tag numbers will be based on the telecom profile default that you defined.

1. Do one of the following:
 - In the **Domain Explorer**, right-click the **Instruments** folder and then click **New**.
 - On the **Instrument Index Module** window menu bar, click **Edit > Tag Numbers > New Tag Number**.
 - On the module toolbar, click **New Tag** .
2. On the **New Tag Number** dialog box, from the **Tag class** list, select **Telecom**.
3. Under **Tag number**, type the name of the new tag number. For details on how to name new tag numbers, see Instrument Tag Naming Convention.

TIP If more than one function identifier exists for the current telecom type acronym, select an appropriate telecom device type on the **Select Telecom Type** dialog box and then click **OK**.
4. On the **New Tag Number** dialog box, click **OK**.
5. On the **Select Instrument Type** dialog box, select the instrument type you want to associate with the current telecom instrument, and then, click **OK**.
6. On the **Tag Number Properties** dialog box, enter appropriate values on the **General** tab.
7. Click the **Telecom** tab and enter your values as needed.
8. If the current field tag requires power supply definitions, click the **Power Supply** tab and enter your values as needed. Make sure that you have selected the **Requires power supply** check box on the **General** tab.
9. Click **OK** to accept all the values that you have entered and to create the field tag.

Duplicate a Telecom Field Tag

This procedure explains how to create a telecom field tag by duplicating the properties of an existing one. You can then edit the new telecom field tag properties as required. You can create the new duplicated telecom field tag in any <unit> of the current domain.

1. In the **Domain Explorer**, double-click the **Instruments** folder to display the existing instruments.
2. Right-click a telecom field tag and then click **Duplicate**.
3. On the **Duplicated Tag Number** dialog box, under **Tag number**, type the name of the new tag number. For details on how to name new tag numbers, see Instrument Tag Naming Convention.
4. To create the new field tag in the another <unit> of the current domain, select the **Create in another unit** check box and do the following:
 - a. On the **Open** dialog box, select the target <unit> for the new field tag.

- b. Click **OK**.
5. Click **OK** to duplicate the field tag and close the **Duplicated Tag Number** dialog box.

TIP In SmartPlant Instrumentation, a device field segment of a telecom field tag, for example, HO, is associated with an existing device type acronym and description, for example, HO HOODS, ACOUSTIC. If you have changed the telecom device field segment in the telecom tag number and the system cannot recognize its association with the existing device field, you must select a telecom device field from the **Select Telecom Device** dialog box, and then click **OK**.
6. On the **Tag Number Properties** dialog box, modify the values as you require and click **OK**.

Create a Telecom Conventional Field Device

This procedure explains how to create a conventional telecom field device. Conventional field devices are not plug- and- socket boxes. Conventional field devices are connected to regular terminals.

You can create a new telecom field device panel in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Field Devices** folder and then on the shortcut menu click **New > Field Device (Conventional)**.
3. On the **Device Panel Properties** dialog box, on the **General** tab, under **Panel**, type a name of the new panel.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Mounting**, type a value if needed.
6. Select the **Set as intrinsically safe** check box if this field device panel has intrinsic safety certification.
7. Click **Revisions** to manage the revisions of the new panel if needed.
8. Click **OK** to accept your definitions for the new field device panel and close this dialog box.

Create a Telecom Plug-and-Socket Field Device

You use plug-and-socket boxes to connect telecom objects that have connectors. The **Plug-and-Socket Box Wizard** enables you to create plug-and-socket boxes or reconfigure the existing ones.

You can create a new telecom plug-and-socket box field device in the **Domain Explorer** or the **Reference Explorer**. If you are creating a plug-and-socket box that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference plug-and-socket boxes facilitate fast creation of plant plug-and-socket boxes by copying required configurations from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a plug-and-socket box that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Field Devices** folder and then on the shortcut menu click **New > Field Device (Plug-and-Socket)**.
3. In the **Plug-and-Socket Box** wizard, follow all the instructions carefully and click **Next** until you complete the wizard.

NOTES

- You can click  next to a list arrow to access the appropriate supporting table. This allows you to add, edit, or delete select list items.
- When defining plug-and-socket box connectors, under **Connector type definition display**, SmartPlant Instrumentation displays the connector sequence, the appropriate pin number, and the pin polarities according to the configuration of the connector than you selected.

Create a Telecom Cable

This procedure explains how to create a telecom cable. When creating a cable, you select an existing cable configuration that includes a particular arrangement of cable sets (pairs, triads, and so forth within a cable) and wires to be built with the cable. You can then add additional cable sets and wires to that cable if required.

If you are creating a cable that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference cables facilitate fast creation of plant cables by copying an existing cable configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a cable that does not have a configuration that you intend to use frequently, we recommend that you create this cable in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the cables in the current <unit>.

1. In the **Domain Explorer**, right-click the **Cables** folder.
2. On the shortcut menu click **New > Cable**.
3. On the **Cable Configuration** dialog box, select an existing telecom cable configuration or create a new one. For details, see *Define a Cable Configuration* (on page 26).
4. Click **Create**.
5. On the **Cable Properties** dialog box, under **Cable**, type a unique cable name.
6. Under **Cable class**, select **Telecom**.
7. Enter the rest of the cable properties as you require and click **OK**.

SECTION 34

Telecom Panels

This section deals with the creation and management of telecom panels.

In This Section

Create a Telecom Junction Box	237
Create a Splice Panel	238
Create a Distribution Frame	238
Create a Patch Panel	239

Create a Telecom Junction Box

This procedure explains how to create a telecom junction box.

You can create a telecom panel in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Junction Boxes** folder and then on the shortcut menu click **New > Telecom Junction Box**.
3. On the **Telecom Junction Box Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new junction box.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the junction box dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this junction box.
TIP Type zero if you want to have an unlimited number of racks.
7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this junction box has intrinsic safety certification.
10. Click the **Associate Symbols** tab to associate a symbol with the current junction box. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new panel if needed.
12. Click **OK** to accept your definitions for the new panel and close this dialog box.

Create a Splice Panel

This procedure explains how to create a splice panel.

You can create a telecom panel in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Splice Panels** folder and then on the shortcut menu click **New > Splice**.
3. On the **Splice Panel Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new panel.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the splice panel dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this splice.
TIP Type zero if you want to have an unlimited number of racks.
7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this splice panel has intrinsic safety certification.
10. Click the **Associate Symbols** tab to associate a symbol with the current splice. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new splice panel if needed.
12. Click **OK** to accept your definitions for the new splice panel and close this dialog box.

Create a Distribution Frame

This procedure explains how to create a distribution frame.

You can create a telecom panel in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Distribution Frames** folder and then on the shortcut menu click **New > Distribution Frame**.

3. On the **Patch Panel Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new panel.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the distribution frame dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this distribution frame.
TIP Type zero if you want to have an unlimited number of racks.
7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this distribution frame has intrinsic safety certification.
10. Click the **Associate Symbols** tab to associate a symbol with the current distribution frame. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new panel if needed.
12. Click **OK** to accept your definitions for the new distribution frame and close this dialog box.

Create a Patch Panel

This procedure explains how to create a patch panel.

You can create a telecom panel in the **Domain Explorer** or the **Reference Explorer**. If you are creating a panel that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference panels facilitate fast creation of plant panels by copying an existing panel configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create a panel that does not have a configuration that you intend to use frequently, we recommend that you create this panel in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Patch Panels** folder and then on the shortcut menu click **New > Patch Panel**.
3. On the **Patch Panel Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new panel.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the patch panel dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this panel.
TIP Type zero if you want to have an unlimited number of racks.
7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this panel has intrinsic safety certification.

10. Click the **Associate Symbols** tab to associate a symbol with the current panel. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new panel if needed.
12. Click **OK** to accept your definitions for the new panel and close this dialog box.

SECTION 35

Equipment Cabinets and Equipment Items

This section deals with the creation and management of equipment cabinets and equipment items.

In This Section

Create a PA Cabinet.....	241
Create an Amplifier.....	242
Add a Connector to an Amplifier.....	243
Add an Amplifier Channel to an Amplifier.....	243
Create a PABX Cabinet.....	244
Create a PABX Rack.....	245
Create a Switch.....	245
Add a Connector to a Switch.....	246
Add a Switch Port.....	246
Create a Switch Channel.....	247
Create and Associate a Telephone Number.....	247
Create an Unassociated Telephone Number.....	248
Associate a Telephone Number with a Channel.....	248
Create a Miscellaneous Cabinet.....	248
Create a Hub Cabinet.....	249
Create Hub Equipment.....	250
Create a Hub Connector.....	250
Add a Port to a Hub.....	251
Create an Intercom Cabinet.....	251
Add Intercom Equipment.....	252
Add an Intercom Connector.....	252

Create a PA Cabinet

This procedure explains how to create a PA cabinet.

SmartPlant Instrumentation allows you to create a number of equipment cabinets that you can use for various purposes. Once you have created the cabinets you require, you can create the appropriate equipment for each cabinet.

You can create an equipment cabinet in the **Domain Explorer** or the **Reference Explorer**. If you are creating an equipment cabinet that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference equipment cabinets facilitate fast creation of plant equipment cabinets by copying an existing cabinet configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an equipment cabinet that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Equipment Cabinets** folder and then on the shortcut menu click **New > PA Cabinet**.
3. On the **PA Cabinet Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new cabinet.

4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the cabinet dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this cabinet.
TIP Type zero if you want to have an unlimited number of racks.
7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this cabinet has intrinsic safety certification.
10. Click the **Associate Symbols** tab to associate a symbol with the current cabinet. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new cabinet if needed.
12. Click **OK** to accept your definitions for the new cabinet and close this dialog box.

Create an Amplifier

The software allows you to add amplifiers to an existing PA cabinet. Once you have created an amplifier, you can create strips and connectors under this amplifier.

SmartPlant Instrumentation allows you to create a number of equipment cabinets that you can use for various purposes. Once you have created the cabinets you require, you can create the appropriate equipment for each cabinet.

You can create an equipment cabinet in the **Domain Explorer** or the **Reference Explorer**. If you are creating an equipment cabinet that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference equipment cabinets facilitate fast creation of plant equipment cabinets by copying an existing cabinet configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an equipment cabinet that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Expand the **Equipment Cabinets** folder and then right-click a PA cabinet to which you want to add an amplifier.
3. On the shortcut menu click **New > Amplifier**.
4. On the **New Wiring Equipment** dialog box, type a name for the new amplifier and click **OK**.
5. On the **Wiring Equipment Properties – Amplifier** dialog box, on the **General** tab, under **Name**, accept or type a name of the new amplifier.
6. Select the **Double width** check box if the amplifier occupies a double width slot or position.
7. In the **Details** group box do the following as you require:
 - a. Type a description.
 - b. Select an amplifier type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.
 - c. Enter a sequence if you need to define the amplifier sequence.
8. Click the **Category Properties** tab.

9. Revise and modify category property values as you require. Click the value for each property and modify it as needed.

TIP Wiring equipment categories that are shipped with SmartPlant Instrumentation have predefined properties. You cannot delete or rename any of these categories or their properties. You can only edit their values. However, you can add user-defined properties to any category which you can rename or delete as you wish. For details, see *Customize and Use Wiring Equipment Categories* (on page 62).

10. Click **OK** to accept your settings and close the dialog box.

Add a Connector to an Amplifier

This procedure explains how to add a connector to an amplifier. In the **SmartPlant Instrumentation Explorer**, connectors are child items of amplifiers and switches.

1. In the **Domain Explorer**, right-click an amplifier to which you want to add a connector.
TIP You can create an amplifier only under a PA cabinet. For more details, see *Create an Amplifier* (on page 242).
2. On the shortcut menu click **New > Connector**.
3. On the **Connector Properties** dialog box, type a connector name.
4. Select a connector type. Click  to add new connector types to this list.
5. Define the connector as male or female.
6. Select a panel side for the connector.
7. Click **OK**.

Add an Amplifier Channel to an Amplifier

This procedure explains how to add an amplifier channel to an amplifier. In the **SmartPlant Instrumentation Explorer**, amplifier channels are child items of amplifiers. You can add as many amplifier channels to an amplifier as you need.

1. In the **Domain Explorer**, right-click an amplifier to which you want to add an amplifier channel.
TIP You can create an amplifier only under a PA cabinet. For more details, see *Create an Amplifier* (on page 242).
2. On the shortcut menu click **New > Amplifier Channel**.
3. On the **New Wiring Equipment** dialog box, under **Name**, type an amplifier channel name and click **OK**.
4. On the **Wiring Equipment Properties** dialog box, define the properties that you require.
5. Click **OK**.

Create a PABX Cabinet

This procedure explains how to create a PABX (private automatic branch exchange) cabinet. Then, you can add PABX racks and terminal strips to the new PABX cabinet.

SmartPlant Instrumentation allows you to create a number of equipment cabinets that you can use for various purposes. Once you have created the cabinets you require, you can create the appropriate equipment for each cabinet.

You can create an equipment cabinet in the **Domain Explorer** or the **Reference Explorer**. If you are creating an equipment cabinet that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference equipment cabinets facilitate fast creation of plant equipment cabinets by copying an existing cabinet configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an equipment cabinet that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Equipment Cabinets** folder and then on the shortcut menu click **New > PABX Cabinet**.
3. On the **PABX Cabinet Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new cabinet.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the cabinet dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this cabinet.
TIP Type zero if you want to have an unlimited number of racks.
7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this cabinet has intrinsic safety certification.
10. Click the **Associate Symbols** tab to associate a symbol with the current cabinet. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new cabinet if needed.
12. Click **OK** to accept your definitions for the new cabinet and close this dialog box.

Create a PABX Rack

This procedure explains how to create a rack in a PABX (private automatic branch exchange) cabinet.

1. In the **Domain Explorer**, expand the **Telecom Panels** folder, which is located within the **Panels by Category** folder.
 2. Expand the **Equipment Cabinets** folder and then right-click a PABX cabinet where you want to create a rack.
 3. On the shortcut menu click **New > PABX Rack**.
 4. On the **Rack Properties** dialog box, on the **General** tab, under **Rack**, type the name of the new rack.
 5. Type a brief description as you require.
 6. Enter the rack sequence in its parent cabinet.
 7. Click **Slots** to add slots to the current rack.
 8. On the **Batch Slot Creation** dialog box, type the number of new slots that you want to add in the new rack.
 9. Do the following to define the slot numbering:
 - a. Type a prefix for the slot name.
 - b. Enter a value in the **Start from number** and **Increment by** fields.
 - c. Click **OK** to add the new slots and to return to the **Rack Properties** dialog box.
- TIP** On the **Rack Properties** dialog box, the software now displays the number of slots that you added.
10. Click the **Associate Symbols** tab to associate a symbol with the current rack. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
 11. Click **OK** to accept your definitions for the new rack and close this dialog box.

Create a Switch

The software allows you to create switches after adding a rack to a PABX cabinet. Once you have created a switch, you can add a switch port and a connector.

1. In the **Domain Explorer**, expand the **Telecom Panels** folder, which is located within the **Panels by Category** folder.
2. Expand the **Equipment Cabinets** folder to display the existing PABX cabinets.
3. Double-click a PABX cabinet to display the existing PABX racks.
4. Right-click a PABX rack to which you want to add a new switch and then on the shortcut menu, click **New > Switch**.
5. On the **New Wiring Equipment** dialog box, type a name for the new switch and click **OK**.
6. On the **Wiring Equipment Properties – Switch** dialog box, on the **General** tab, under **Name**, accept or type a name of the new switch.
7. Select the **Double width** check box if the amplifier occupies a double width slot or position.
8. In the **Details** group box do the following as you require:
 - a. Type a description.
 - b. Select an amplifier type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.
 - c. Enter a sequence if you need to define the switch sequence.

9. Click the **Category Properties** tab.
10. Revise and modify category property values as you require. Click the value for each property and modify it as needed.
TIP Wiring equipment categories that are shipped with SmartPlant Instrumentation have predefined properties. You cannot delete or rename any of these categories or their properties. You can only edit their values. However, you can add user-defined properties to any category which you can rename or delete as you wish. For details, see *Customize and Use Wiring Equipment Categories* (on page 62).
11. Click **OK** to accept your settings and close the dialog box.

Add a Connector to a Switch

This procedure explains how to add a connector to a switch. In the **SmartPlant Instrumentation Explorer**, connectors are child items of switches and amplifiers.

1. In the **Domain Explorer**, expand the **Telecom Panels** folder, which is located within the **Panels by Category** folder.
2. Expand the **Equipment Cabinets** folder to display the existing PABX cabinets.
3. Double-click a PABX cabinet and then a PABX rack to display the existing switches.
4. Right-click a switch to which you want to add a new connector and then on the shortcut menu, click **New > Connector**.
5. On the **Connector Properties** dialog box, type a connector name.
6. Select a connector type. Click  to add new connector types to this list.
7. Define the connector as male or female.
8. Select a panel side for the connector.
9. Click **OK**.

Add a Switch Port

This procedure explains how to add a port to an existing switch.

1. In the **Domain Explorer**, expand the **Telecom Panels** folder, which is located within the **Panels by Category** folder.
2. Expand the **Equipment Cabinets** folder to display the existing PABX cabinets.
3. Double-click a PABX cabinet and then a PABX rack to display the existing switches.
4. Right-click a switch to which you want to add a new port and then on the shortcut menu, click **New > Switch Port**.
5. On the **General** tab of the **Switch Port Properties** dialog box, type a name for the new switch port.
6. Click the **Associate Symbols** tab to associate a symbol with the current switch port. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
7. Click **OK**.

Create a Switch Channel

After creating a switch port, you can add a channel to this port.

1. In the **Domain Explorer**, expand the **Telecom Panels** folder, which is located within the **Panels by Category** folder.
2. Expand the **Equipment Cabinets** folder to display the existing PABX cabinets.
3. Expand the hierarchy of a PABX cabinet and then double-click a PABX rack, switch, and then a switch port.
4. Right-click a switch port to which you want to add a new channel and then on the shortcut menu, click **New > Switch Channel**.
5. On the **General** tab of the **Channel Properties** dialog box, type a name for the new channel.
6. Type a brief description if required.
7. Enter an appropriate sequence of the current channel in its parent switch port.
8. Click the **Associate Symbols** tab to associate a symbol with the current channel. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
9. Click **OK** to accept your settings and close this dialog box.

Create and Associate a Telephone Number

This option allows you to create a telephone number in a PABX cabinet and automatically associate it with a switch channel.

1. In the **Domain Explorer**, expand the **Telecom Panels** folder, which is located within the **Panels by Category** folder.
2. Expand the **Equipment Cabinets** folder to display the existing PABX cabinets.
3. Expand the hierarchy of a PABX cabinet and then double-click a PABX rack, switch, and then a switch channel.
4. Right-click a switch channel and create a terminal if there is no terminal under the current switch channel.
5. Right-click the **Telephone** folder and then on the shortcut menu, click **New > Telephone**.
6. On the **Telephone Properties** dialog box, type a telephone number and description as you require.
7. Select an appropriate telephone state and usage. If the values that you need do not exist on the **Telephone state** and **Telephone usage** lists, you can add them in the Telecom supporting tables in the Wiring module. To do this, in the Wiring module, click **Tables > Telecom > Telephone State** or **Telephone Usage**.
8. Click **OK**.

Create an Unassociated Telephone Number

1. In the Wiring module, click **Tables > Telecom Telephone Numbers**.
2. On the **Telephone Numbers** dialog box, click **New**.
3. Type an appropriate number, description, telephone state, and telephone usage as you require.
4. Click **OK**.

Associate a Telephone Number with a Channel

The software allows you to associate a telephone number with a switch channel. Note that a switch channel can be associated with many telephone numbers.

1. In the **Domain Explorer**, expand the hierarchy of a PABX panel to display the existing racks, switches, switch ports, and terminals.
2. Right-click a channel, then on the shortcut menu, click **Actions > Associate Telephone Numbers**.
3. On the **Associate Telephone Numbers** dialog box, select an available telephone number and click **Associate**.
4. Click **Close** when done.

Create a Miscellaneous Cabinet

Miscellaneous cabinets are intended for video equipment, entertainment systems, and so forth. After creating a miscellaneous cabinet, you can add a rack and a terminal strip.

SmartPlant Instrumentation allows you to create a number of equipment cabinets that you can use for various purposes. Once you have created the cabinets you require, you can create the appropriate equipment for each cabinet.

You can create an equipment cabinet in the **Domain Explorer** or the **Reference Explorer**. If you are creating an equipment cabinet that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference equipment cabinets facilitate fast creation of plant equipment cabinets by copying an existing cabinet configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an equipment cabinet that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Equipment Cabinets** folder and then on the shortcut menu click **New > Miscellaneous Cabinet**.
3. On the **Miscellaneous Cabinet Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new cabinet.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the cabinet dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this cabinet.

TIP Type zero if you want to have an unlimited number of racks.

7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this cabinet has intrinsic safety certification.
10. Click the **Associate Symbols** tab to associate a symbol with the current cabinet. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new cabinet if needed.
12. Click **OK** to accept your definitions for the new cabinet and close this dialog box.

Create a Hub Cabinet

This procedure explains how to create a hub cabinet. After creating a hub cabinet, you can add hub equipment.

SmartPlant Instrumentation allows you to create a number of equipment cabinets that you can use for various purposes. Once you have created the cabinets you require, you can create the appropriate equipment for each cabinet.

You can create an equipment cabinet in the **Domain Explorer** or the **Reference Explorer**. If you are creating an equipment cabinet that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference equipment cabinets facilitate fast creation of plant equipment cabinets by copying an existing cabinet configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an equipment cabinet that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Equipment Cabinets** folder and then on the shortcut menu click **New > Hub Cabinet**.
3. On the **Hub Cabinet Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new cabinet.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the cabinet dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this cabinet.

TIP Type zero if you want to have an unlimited number of racks.
7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this cabinet has intrinsic safety certification.
10. Click the **Associate Symbols** tab to associate a symbol with the current cabinet. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new cabinet if needed.
12. Click **OK** to accept your definitions for the new cabinet and close this dialog box.

Create Hub Equipment

You can add hub equipment to a hub cabinet. Hub equipment is a type of wiring equipment. After creating hub equipment, you can add a port and a hub connector.

1. In the **Reference Explorer** or **Domain Explorer**, expand the **Telecom Panels** folder.
2. Expand the **Equipment Cabinets** folder to display the existing hub cabinets.
3. Right-click a hub cabinet and then on the shortcut menu, click **New > Hub Equipment**.
4. On the **New Wiring Equipment** dialog box, type a name for the new hub equipment and then click **OK**.
5. On the **General** tab of the **Wiring Equipment Properties – Hub** dialog box, type the name of the new hub.
6. Select the **Double width** check box if the hub occupies a double width slot or position.
7. In the **Details** group box do the following as you require:
 - a. Type a description.
 - b. Select a hub type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.
 - c. Enter a sequence if you need to define the hub sequence.
8. Click the **Category Properties** tab.
9. Revise and modify category property values as you require. Click the value for each property and modify it as needed.

TIP Wiring equipment categories that are shipped with SmartPlant Instrumentation have predefined properties. You cannot delete or rename any of these categories or their properties. You can only edit their values. However, you can add user-defined properties to any category which you can rename or delete as you wish. For details, see *Customize and Use Wiring Equipment Categories* (on page 62).
10. Click **OK** to accept your settings and close the dialog box.

Create a Hub Connector

This procedure explains how to add a hub connector to a hub.

1. In the **Domain Explorer** or **Reference Explorer**, expand a hub cabinet hierarchy to display the existing hub equipment and hub ports.
2. Do one of the following:
 - Right-click a port to which you want to add a hub connector.
 - Right-click the **Hub Connectors** folder.
3. On the shortcut menu, click **New > Hub Connector**.
4. On the **Connector Properties** dialog box, type a connector name.
5. Select a connector type. Click  to add new connector types to this list.
6. Set whether the connector is male or female.
7. Select a panel side for the connector.
8. Click **OK**.

Add a Port to a Hub

This procedure explains how to add a port to a hub.

1. In the **Domain Explorer** or **Reference Explorer**, navigate to a hub cabinet.
2. Double-click a hub cabinet to display the existing hub equipment.
3. Right-click a hub equipment item and then on the shortcut menu, click **New > Port**.
4. On the **General** tab of the **Port Properties** dialog box, type a name for the new port.
5. Type a brief description if necessary.
6. Enter the port sequence in the hub if needed.
7. Click the **Associate Symbols** tab to associate a symbol with the current port. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
8. Click **OK**.

Create an Intercom Cabinet

This procedure explains how to create an intercom cabinet. After creating an intercom cabinet, you can add intercom equipment and a terminal strip.

SmartPlant Instrumentation allows you to create a number of equipment cabinets that you can use for various purposes. Once you have created the cabinets you require, you can create the appropriate equipment for each cabinet.

You can create an equipment cabinet in the **Domain Explorer** or the **Reference Explorer**. If you are creating an equipment cabinet that is going to have a frequently used configuration, we recommend that you create it in the **Reference Explorer**. Reference equipment cabinets facilitate fast creation of plant equipment cabinets by copying an existing cabinet configuration from the **Reference Explorer** to the **Domain Explorer**. However, if you want to create an equipment cabinet that does not have a configuration that you intend to use frequently, we recommend that you create it in the **Domain Explorer** and in the <unit> where it is to be physically located. This is helpful when filtering the panels in the current <unit>.

1. In the **Domain Explorer**, double-click the **Panels by Category** folder and expand the **Telecom Panels** folder.
2. Right-click the **Equipment Cabinets** folder and then on the shortcut menu click **New > Intercom Cabinet**.
3. On the **Intercom Cabinet Properties** dialog box, on the **General** tab, under **Panel**, type the name of the new cabinet.
4. Select the required values from the **Type**, **Manufacturer**, **Model**, **Area classification**, and **Location** lists. If the required value is not available, click  next to the relevant list arrow to open the appropriate supporting table.
5. Under **Dimensions**, type a value for the cabinet dimensions if needed.
6. Under **Maximum number of racks**, type a value to set the maximum number of racks that will be possible to create in this cabinet.

TIP Type zero if you want to have an unlimited number of racks.
7. Under **Mounting**, type a value if needed.
8. Under **Backplane**, type a value if needed.
9. Select the **Set as intrinsically safe** check box if this cabinet has intrinsic safety certification.

10. Click the **Associate Symbols** tab to associate a symbol with the current cabinet. The software will use this symbol when generating a report in the Enhanced Report Utility. For details, see *Associate a Symbol with an Item* (on page 23).
11. Click **Revisions** to manage the revisions of the new cabinet if needed.
12. Click **OK** to accept your definitions for the new cabinet and close this dialog box.

Add Intercom Equipment

This procedure explains how to add intercom to an intercom cabinet.

1. In the **Reference Explorer** or **Domain Explorer**, expand the **Telecom Panels** folder.
2. Expand the **Equipment Cabinets** folder to display the existing intercom cabinets.
3. Right-click an intercom cabinet and then on the shortcut menu, click **New > Intercom Equipment**.
4. On the **New Wiring Equipment** dialog box, type a name for the new intercom equipment and then click **OK**.
5. On the **General** tab of the **Wiring Equipment Properties – Intercom** dialog box, type the name of the new intercom.
6. Select the **Double width** check box if the hub occupies a double width slot or position.
7. In the **Details** group box do the following as you require:
 - a. Type a description.
 - b. Select a intercom type, model, and manufacturer. If the required value is not available on the list, click  to define a new one.
 - c. Enter a sequence if you need to define the intercom sequence in the cabinet.
8. Click the **Category Properties** tab.
9. Revise and modify category property values as you require. Click the value for each property and modify it as needed.

TIP Wiring equipment categories that are shipped with SmartPlant Instrumentation have predefined properties. You cannot delete or rename any of these categories or their properties. You can only edit their values. However, you can add user-defined properties to any category which you can rename or delete as you wish. For details, see *Customize and Use Wiring Equipment Categories* (on page 62).
10. Click **OK** to accept your settings and close the dialog box.

Add an Intercom Connector

This procedure explains how to a connector to an intercom.

1. In the **Domain Explorer** or **Reference Explorer**, expand a hub cabinet hierarchy to display the existing intercom equipment.
2. Do one of the following:
3. Right-click an intercom to which you want to add a connector.
4. On the shortcut menu, click **New > Connector**.
5. On the **Connector Properties** dialog box, type a connector name.
6. Select a connector type. Click  to add new connector types to this list.
7. Set whether the connector is male or female.
8. Select a panel side for the connector.
9. Click **OK**.

Loop Drawings Module

You use the Loop Drawings module to retrieve predefined instrumentation data and generate appropriate loop drawings. Each component in the loop drawing is assigned the necessary drawing blocks that represent its functionality and wiring routing. The drawings display information about the loops and their instruments, wiring routing, line data, DCS data, document references, approvals, revisions, and general information.

The Loop Drawings module generates loop drawings by means of a fast build process. By configuring typical drawing blocks for key instrument types, the loop components can be automatically initialized with their corresponding graphical elements. The Loop Drawings module enables you to create loop drawings that include data retrieved from the database and if required, from an external file.

You can generate loop drawings in one of the following ways:

- Using the Enhanced Report Utility.
- Using an external CAD application (SmartPlant Instrumentation supports Intergraph SmartSketch, Autodesk AutoCAD, or Bentley Systems MicroStation for Windows).
 - **NOTE** SmartPlant Instrumentation and the CAD drawing programs required to work with SmartPlant Instrumentation, must be installed on the same machine.

CAD Drawing Blocks in SmartPlant Instrumentation

In SmartPlant Instrumentation, a block is considered as a template CAD drawing file (with or without macros) that is assigned to a loop or instrument. The software uses blocks in CAD loop drawing generation. You can use similar blocks to represent identical components in different drawing positions by defining several blocks that use the same drawing file.

A block is a term that is used in SmartPlant Instrumentation. Depending on the CAD application that you use, the following equivalent terms apply.

SmartPlant Instrumentation Term	SmartSketch	AutoCAD	MicroStation
Block	Symbol	Drawing	Cell
Macro	Text element	Attribute element	Tag

There must be a correlation between the file formats supported by your CAD application and the file format in which you save a new drawing block. Standard drawing blocks exist for each of the following CAD applications (shown with the appropriate file extension):

- Intergraph SmartSketch — .sym
- AutoCAD — .dwg
- MicroStation — .cel

SmartPlant Instrumentation is shipped with several examples of blocks that are used in the demonstration database (IN_DEMO.DB). The wiring methodology behind these blocks includes the use of an overall shield with single cables and multiple pair cables. After the installation, these blocks can be found in the path <SmartPlant Instrumentation home folder>\CAD\Blocks. For details, see *Shipped Block Files* (on page 257).

Block Organization and Management

SmartPlant Instrumentation requires that each drawing block belongs to a particular block type. Therefore, after you define block files in your CAD application, you need to assign these blocks to the appropriate block types in SmartPlant Instrumentation.

You define and manage blocks in the **Drawing Block Types** folder of the **Domain Explorer**. After you define blocks and organize them according to their block types, you can perform block association procedures, which depend on the block type of the source block. Blocks that are already associated with specific loops or instruments appear in the **Loops** folder of the **Domain Explorer**. For details of the available block types, see *Block Types* (on page 254).

Block Assignment Methods

After defining the blocks and their macros, it is necessary to assign your instrument tag numbers to the drawing blocks that represent the instruments graphically. The following is the description of the available assignment methods:

- Automatic block assignment method — In the **Loops** folder of the **Domain Explorer**, these blocks are shown under instruments as . Also, you must use this method if you want to assign user-defined general blocks to your loop drawings. After assigning such a block to an instrument type, the software displays this block in every loop drawing whose source loop contains an instrument with the same instrument type. The assigned user-defined general blocks do not appear in the **Domain Explorer**.
- Manual block assignment method — In the **Loops** folder of the **Domain Explorer**, these blocks are shown under instruments as . In the **Domain Explorer**, it is possible to change the block assignment method from automatic to manual.

Block Types

SmartPlant Instrumentation requires each drawing block to belong to a specific block type. This is because blocks in SmartPlant Instrumentation function according to the block type they are assigned to. For example, if a block belongs to the **Loop Block** type, you can only assign this block to a loop; if a block belongs to an Instrument Block type, you can only assign this block to a specific instrument of the loop but not to the loop itself.

You organize blocks according to block types in the **Drawing Block Types** folder of the **Domain Explorer**. At this level, the software displays all the block types that exist in the current domain and allows you to add general and instrument block types and manage block type properties. All block types that appear in the current <plant> also appear in all <plants> that exist in the domain.

The following drawing block types are available in the **Drawing Block Types** folder of the **Domain Explorer**:

- **Loop Block** () — A shipped block type that allows you to define, edit or duplicate a block that you can associate with a loop number. The **Loop Block** folder is the only folder of the **Domain Explorer** where you can define and manage loop blocks. Loop blocks are shown as . After defining a loop block, you can assign it directly to a loop number in the **Loops** folder of the **Domain Explorer**. When generating a CAD loop drawing using a loop block, the generation is based on a single template block file that can include references to all tags in the loop. For more details, see *Loop Blocks* (on page 256).
- **Instrument Block** () — A user-defined block type that allows you to define, edit, or duplicate a block that you can associate with an instrument. Instrument blocks are shown as . Blocks that you assign to instruments appear at the instrument level in the **Loops** folder of the **Domain Explorer**. When generating a CAD loop drawing using instrument blocks, the generation is based on template CAD blocks that contain macros specific to each instrument tag of the loop. This requires assignment of individual drawing block files to instruments whose data you want to display in the loop drawing (one block file per instrument). In the generated drawing, the software displays the instrument blocks according to their insertion points that you defined. The macros the blocks contain retrieve data from SmartPlant Instrumentation for all the associated instruments. You can create as many instrument block types as you require. After defining instrument blocks, you can choose a block assignment method to associate your blocks with specific instruments; therefore, you must define at least one instrument block type before you can add a new block (apart from the default general block types and the **Loop Block** type).
- **Border** () — A shipped block type that allows you to define a border block for use in all loop drawings or hook-up drawings. You can also modify properties of an existing loop or hook-up drawing border block. Border blocks are shown as . After defining border blocks, you can assign one border block to all loop drawings or hook-up drawings using the **Default General Blocks** dialog box.
- **Logo** () — A shipped block type that allows you to define a logo block for use in all loop drawings or hook-up drawings. You can also modify properties of an existing loop or hook-up drawing logo block. Logo blocks are shown as . After defining logo blocks, you can assign one logo block to all loop drawings or hook-up drawings using the **Default General Blocks** dialog box.
- **User-Defined General Block** () — A user-defined block type that allows you to define, edit, or duplicate a block that you want to use in all loop drawings in addition to the border and logo blocks. This block does not contain any macros. User-defined general blocks are shown as . You can assign any number of user-defined general blocks to loop instruments using the Automatic block assignment method. The software automatically displays all the assigned user-defined general blocks in every CAD loop drawing that you generate, regardless of the CAD application you use. User-defined general blocks cannot be assigned to hook-up drawings.

NOTES

- The software always places in the drawing one block belonging to the **Logo** type and one block belonging to the **Border** type regardless of whether you add user-defined general blocks in the drawing. For a complete list of the shipped logo and border blocks, see *Shipped Block Files* (on page 257).
- You can specify the coordinates of user-defined general blocks to prevent overlapping with other drawing elements, such as the border or the logo.
- In hook-up drawings, you can only use the blocks created in the default **Border** and **Logo** general block types.

Loop Blocks

A loop block is a CAD drawing block with macros that retrieve data from all instrument tags of a specific loop. Since a loop block is a single template block file, you do not need to specify insertion points for this block. You need to create loop blocks in your CAD application and modify the macros as appropriate. Then, in SmartPlant Instrumentation, in the **Drawing Block Types > Loop Block** folder of the **Domain Explorer**, you define loop blocks you want to make available for association with loop numbers. After that, you can associate the loop blocks with loop numbers displayed in the **Loops** folder of the **Domain Explorer**.

You define loop blocks and manage their properties in the same way as you do with instrument blocks. You can only assign one loop block to a particular loop. However, in contrast to instrument blocks, you cannot assign the same loop block to more than one loop. Therefore, when duplicating a loop that already has a loop block assigned, the software ignores the loop block for the duplicated loop.

In loop drawing generation, the loop block has the highest generation priority. The software ignores the instrument blocks when generating a loop drawing if the source loop has a loop block and also blocks assigned to the loop instruments via the instrument type using the automatic block assignment method.

The loop block macro structure is **MACRO.<macro function>.<instrument type>.<tag suffix>**, where each segment of the macro structure is described as follows:

- **MACRO** — This segment denotes an existing macro used by SmartPlant Instrumentation in CAD loop drawing generation. A loop block macro definition can be either **MACRO**, **MACRO.X**, or **MACRO.X.Y** if the macro requires additional parameters.
- **Macro function** — This is an optional segment displaying the macro function name, for example, **TRIM**.
- **Instrument type** — This is a required segment that denotes the instrument type abbreviation of a loop tag, for example, **FT**, **FE**, and so forth.
- **Tag suffix** — This segment denotes the suffix identification defined in the instrument naming convention. This segment is only required if a loop includes more than one instrument with the same instrument type.

The following are examples of loop block macros:

- **TAG_MOD_DESC.FE** — The instrument model description for the flow element in the loop.
- **TERM_NUM.1.3.FT.B** — The terminal name of the first termination of the wire level 3 for the second (Suffix B) flow transmitter in the loop.
- **TAG_NUM.s54.LSHH** — Four characters of the level switch name starting from the fifth character, as defined in the **s54** macro function. For example, if the tag number is 101-LSHH -100, the macro retrieves **LSHH**.

Shipped Block Files

Border and Logo Block Files

The files in the list are the predefined drawing blocks that you can use to represent the border / title (one of which should be defined as the default border) and the logo (one of which should be defined as the default logo).

File Name	Description	Available Formats
BORD_DEF	Border and title.	.dwg, .sym
BORD_DEM	Border and title.	.dwg
BORDER	Border and title.	.cel
BRDDEM	Border and title.	.cel
TITLE	Border and title.	.cel
INGR	Intergraph logo.	.cel
LOGO_DEF	Intergraph logo.	.dwg, .sym
LOGO_DEM	Intergraph logo.	.dwg, .sym
LOGO_IN	Intergraph logo.	.dwg
LOGOIG	Intergraph logo.	.cel

DCS and Control Block Files

File Name	Description	Available Formats
CTRL1	Analog input control.	.dwg
CTRL2	Analog input and analog output control.	.dwg

Field Device Block Files

File Name	Description	Available Formats
CV2	Control valve.	.dwg, .sym
ORIF_1	Orifice on a steam traced line.	.dwg
ORIF_2	Orifice.	.dwg
PLOC_IND.DWG	Parallel connection of an indicator.	.dwg
PT_1	Pressure element.	.dwg
SLOC_IND.DWG	Series connection of an indicator.	.dwg
TC_1	Skin temperature controller (welded).	.dwg
TC_2	Temperature element.	.dwg

File Name	Description	Available Formats
TE	Temperature element wired to a temperature transmitter.	.dwg

Wiring Routing Block Files

File Name	Description	Available Formats
AIT_ROUT	Typical analog input routing; field device, junction box, marshaling rack, and an analog input I/O card.	.dwg
AOT_ROUT	Typical analog output routing; I/P transducer, junction box, marshaling rack, and an analog output I/O card.	.dwg
PAIT1	Use for parallel indicator and transmitter.	.dwg
SAIT	Use for serial indicator and transmitter.	.dwg

Managing CAD Drawing Blocks Common Tasks

The following tasks are used when you create and manage CAD drawing blocks.

Add a Block Type

Use this procedure to add a new instrument block type or a user-defined general block type in the **Drawing Block Types** folder of the **Domain Explorer**.

For more information, see *Add a Block Type* (on page 259).

Add a Block to a Block Type

Use this procedure to add a block to a block type.

For more information, see *Add a Block to a Block Type* (on page 259).

View a Drawing Block in a CAD Application

Use this procedure to display a block in your CAD application — SmartSketch, MicroStation or AutoCAD — depending on the settings you made on the **Preferences** dialog box.

For more information, see *View a Drawing Block in a CAD Application* (on page 260).

Define Insertion Points for Blocks

In SmartSketch, AutoCAD or MicroStation, you can define the insertion point for individual blocks relative to the drawing origin (lower left corner). This enables you to use the same block in different positions in different loop drawings.

For more information, see *Define Insertion Points for Blocks* (on page 260).

Copy Blocks to Another <Plant>

Use this procedure to copy one or more blocks that you select in the **Drawing Block Types** folder of the **Domain Explorer** to another <plant> that exists in the current domain.

For more information, see *Copy Blocks to Another <Plant>* (on page 261).

Add a Block Type

1. In the **Domain Explorer**, right-click the **Drawing Block Types** folder and, on the shortcut menu, click **New**.
2. Do one of the following:
 - Click **General Block Type** to create a user-defined general block type.
 - Click **Instrument Block Type** to create a block type whose blocks you can then assign to instrument tags in the **Loops** folder of the **Domain Explorer**.
3. In the **BlockType Properties** dialog box, type the new block name and description in the appropriate boxes.

TIPS

- Block types are defined at the domain level; therefore, a block type that you add in the current <plant> automatically appears in all the <plants> that exist in the current domain.
- You cannot add a new block type for loop blocks.

Add a Block to a Block Type

1. In the **Domain Explorer**, double-click the **Drawing Block Types** folder to expand its hierarchy.
2. Right-click a block type and on the shortcut menu click **New > Block**.
3. On the **Block Properties** dialog box, enter information as follows:
 - a. In the **Block** field, enter the name to be used for the block by SmartPlant Instrumentation.
 - b. In the **Description** field, type in a description for the block.
 - c. Under **Block type**, select a block type from the list.
4. Under **Drawing block file**, navigate to the block source file by clicking **Browse**.
5. Specify the file path.
6. Under **Origin**, enter the required X and Y parameters to position the new block on the drawing.

TIPS

- You can enter decimal numbers as well as whole numbers.

- Blocks that you add under the **Border**, **Logo**, or a user-defined general block type appear on the **Default General Blocks** dialog box, which you open from the Actions menu of the **Loop Drawings Module** window menu bar. In the **Domain Explorer**, general blocks only appear in the **Drawing Block Types** folder and are shown as .
- Loop blocks that you add become available for association with loops in the **Loops** folder of the **Domain Explorer**. These blocks are shown as .
- Instrument blocks that you add become available for association with instrument tag numbers in the **Loops** folder of the **Domain Explorer**. In the **Drawing Block Types** folder, instrument blocks are shown as . In the **Loops** folder, blocks assigned to instruments using the automatic block assignment method are shown as  and blocks assigned to instruments using the manual block assignment method are shown as .

View a Drawing Block in a CAD Application

NOTE The difference between viewing and generating is that viewing only shows the code fields of blocks and macros while generating actually retrieves the data and displays it on the drawing layer.

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the tree view pane, expand the loop hierarchy to the tag level and select a tag.
3. In the **Items** pane, right-click a block.
4. On the shortcut menu, click **Reports > View Block in CAD Application**.

Define Insertion Points for Blocks

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. Expand the loop-tag-block hierarchy and then do one of the following:
 - In the tree view pane, right-click a desired block.
 - In the **Items** pane, select and right-click several blocks.

TIP You can only define insertion points for blocks shown as . These blocks are assigned to instruments using the manual block assignment method. If your block selection in the **Items** pane includes automatically assigned blocks, (shown as ) , you need to redefine your selection.

3. On the shortcut menu, click **Actions > Define Block Origin**.
4. On the **Define Block Origin** dialog box, select a block.

TIP If there is a large number of blocks in the data window, under **Find**, select a column heading and then type a value. For example, if you select **Tag Number** as a column heading and type a specific tag number, in the data window, the software finds the block assigned to this tag number.

5. Under **Tag assignments**, enter the required values (in the CAD application units) for the block new insertion point X- and Y-coordinates.

TIP The block origin can consist of numbers that include decimal values, for example, 9.85.

Copy Blocks to Another <Plant>

1. In the **Domain Explorer**, double-click the **Drawing Block Types** folder to expand its hierarchy.
2. Double-click a block type whose blocks you want to copy.
3. Do one of the following:
 - In the tree view pane, right-click a specific block.
 - In the **Items** pane, select and right-click blocks you want to copy.
4. On the shortcut menu, click **Actions > Copy Blocks to Another <Plant>**.
5. In the **Items** pane, select and right-click the blocks that you want to copy to other tag numbers.
6. On the shortcut menu, click **Actions > Copy Blocks to Tag Numbers**.
7. On the dialog box that opens, select a target plant and click **OK**.

TIPS

- The blocks that you copy appear in the appropriate block type of the target plant. The block types you define in the source plant automatically appear in all the plants in the current domain because block types are defined at the domain level.
- When copying blocks, the software does not copy any associations you made for the blocks in the source plant.

CAD Drawing Block Assignment Common Tasks

The following tasks are used when you assign blocks to instrument loops or tags.

Associate a Loop Block

Use this procedure to associate a loop block with one or more loop numbers. You can only assign a loop block if the loop that you selected does not have any instrument blocks assigned using the manual block assignment method. Such blocks are shown under the loop tags as . The software, however, does not prevent you from assigning a loop block to the loop if the loop tags have instrument blocks assigned using the automatic block assignment method. Such blocks are shown under the loop tags as . After associating a block with a loop, when generating a loop drawing, the software always uses the loop block and ignores instrument blocks this loop might contain. For more information, see *Associate a Loop Block* (on page 263).

Associate Instrument Blocks Automatically (via Instrument Type)

In an instrumentation design setup with a very large number of control loops, similar tag numbers are represented repeatedly using the same drawing block, the only difference being the data that is retrieved by the macros in the drawing. Therefore, instead of having to associate each block with an instrument manually, you can use this procedure to associate typical blocks with a specific instrument type. For more information, see *Associate Instrument Blocks Automatically (via Instrument Type)* (on page 263).

Change the Block Assignment Method from Automatic to Manual

This procedure enables you to change the assignment method of instrument blocks from automatic to manual. Blocks assigned to tag numbers using the manual block assignment method become available for selection in the **Generate Loop Drawings** dialog box, where you can set the software to generate loop drawings only for those loops that use a specific manually assigned instrument block. For more information, see *Change the Block Assignment Method from Automatic to Manual* (on page 264).

Associate Instrument Blocks Manually

You can manually associate one or more blocks with tag numbers belonging to the same loop number. In this case, that the software draws blocks after you select a loop number in the **Domain Explorer**. For more information, see *Associate Instrument Blocks Manually* (on page 265).

Update Block Assignments

This procedure enables you to update block assignments that might have been changed during your <plant> life cycle. You can update block assignments in batch mode, for one or more instrument tags belonging to the same loop or to a group of loops. For more information, see *Update Block Assignments* (on page 265).

Copy Associated Blocks to Other Instrument Tags

Use this procedure to copy one or more blocks that are associated with a particular instrument to other instrument tags displayed in the **Domain Explorer**. For more information, see *Copy Associated Blocks to Other Instrument Tags* (on page 266).

Copy Automatically Assigned Blocks to Tags of Other Loops

Use this procedure to copy blocks from all instrument tags of a specific loop to all instruments of one or more target loops. The software only copies those blocks that are assigned to the source loop instruments using the automatic block assignment method. In the **Domain Explorer**, these blocks are indicated with the  icon. The software validates the instrument tag assignments and the drawing type of the target loops before copying the blocks. For more information, see *Copy Automatically Assigned Blocks to Tags of Other Loops* (on page 266).

Assign an Instrument Block to a Drawing Page

Use this procedure to assign a CAD drawing block to a specific page in the generated loop drawing when the loop drawing multi-page drawing or multi-drawing. You must perform this procedure if the drawing block is assigned to the loop instrument using the automatic block assignment method. Such blocks are shown under the loop tags as . For more information, see *Assign an Instrument Block to a Drawing Page* (on page 267).

Dissociate Blocks from Instruments

Use this procedure to dissociate blocks from instruments belonging to the same or different loops. For more information, see *Dissociate Blocks from Instruments* (on page 267).

Specify Border and Logo Blocks for Loop Drawings

This procedure allows you to set SmartPlant Instrumentation to use a specific border block and logo block every time you generate a CAD loop drawing. Blocks that you set as default general blocks must belong to the shipped general block types, that is, Border and Logo. You define and manage blocks in the **Drawing Block Types** folder of the **Domain Explorer**. For more information, see *Specify Border and Logo Blocks for Loop Drawings* (on page 268).

Assign General Blocks to All Loop Drawings via Instrument Type

Use this procedure to assign user-defined general blocks to a specific instrument type. This procedure allows you to display general blocks in all loop drawings automatically if the source loops have tags belonging to the instrument type to which you assigned the blocks. The software displays such blocks in addition to the default border and logo blocks that you specify on the **Default General Blocks** dialog box. For more information, see *Assign General Blocks to All Loop Drawings via Instrument Type* (on page 268).

Associate a Loop Block

NOTE You must first define loop blocks in the **Drawing Block Types** folder of the **Domain Explorer**. In the **Domain Explorer**, loop blocks are shown as . For details, see *Add a Block to a Block Type* (on page 259).

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the tag level in the **Loops** folder.
2. Do one of the following:
 - In the tree view pane, right-click a specific loop number.
 - In the **Items** pane, select and right-click several loop numbers.
3. On the shortcut menu, click **Actions > Associate Loop Block**.

TIPS

- If there is a large number of blocks in the data window, under **Find block**, select a column heading and then type a value. For example, if you select **Block** as a column heading and type block name **Control**, in the data window, the software selects the block whose name starts with **Control**.
 - If you want to display only specific blocks in the data window, click **Filter** and specify a filter parameter.
4. In the data window, select the loop block and then click **OK**.

TIPS

- You can only assign one loop block per loop.
- To dissociate loop block, right-click it and then, on the shortcut menu, click **Actions > Dissociate Loop Block**.

Associate Instrument Blocks Automatically (via Instrument Type)

1. In the **Loop Drawings Module** window, do one of the following:
 - On the toolbar, click .
 - Click **Actions > Block - Instrument Type Assignment**.

2. On the **Block - Instrument Type Assignment** dialog box, in the **Type** data window column, select an instrument type.

TIP If there is a large number of instrument types in the data window, under **Find instrument type**, select a column heading and then type a value. For example, if you select **Description** as a column heading and type **Control**, in the data window, the software locates the instrument types whose description starts with **Control**.

3. Click **Assign**.
4. If you defined several instrument block types in the **Drawing Block Types** folder of the **Domain Explorer**, from the **Block type** list, select the appropriate instrument block type.
5. Select blocks that you want to assign to the instrument type.

TIPS

- If you select **Show blocks of all types**, the software displays all blocks belonging to all the instrument block types that you defined in the **Drawing Block Types** folder of the **Domain Explorer**. In addition to instrument blocks, the software displays all user-defined general blocks. If you assign a user-defined blocks to the instrument type, the software displays this block in all loop drawings that include an instrument belonging to the current instrument type.
- Click **View** to open a block in your CAD application.

6. Click **OK** and then check the block assignment in the **Loops** folder of the **Domain Explorer**.

TIP In the **Loops** folder of the **Domain Explorer**, the instrument blocks that you assigned appear under the tag numbers belonging to the instrument type that you selected. These blocks are indicated with the icon .

Change the Block Assignment Method from Automatic to Manual

NOTE You cannot use this procedure if the source loop contains a loop block (shown as  under the loop number).

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the instrument level in the **Loops** folder.
2. Do one of the following:
 - In the **Items** pane, select and right-click one or more loops.
 - In the tree view pane, select a specific loop and then in the **Items** pane, select and right-click instruments whose blocks are assigned using the automatic block assignment method.

TIP In the **Loops** folder, blocks associated with tags using the manual method are marked with the icon . Blocks associated with tags using the automatic block assignment method are marked with the icon .

3. On the shortcut menu, click **Actions > Change Block Assignment Method**.
4. Click **Yes** when prompted.

TIPS

- In the **Domain Explorer**, the block icons change accordingly.
- After changing the block assignment method from automatic to manual, these blocks are no longer associated with an instrument type, and therefore are not affected by any changes you or other users can make to the block - instrument type association.

- Manually associated blocks become available for selection in the **Generate Loop Drawings** dialog box, where you can generate a CAD loop drawing per block.

Associate Instrument Blocks Manually

NOTE If you have a large number of loops with tag numbers that have identical properties, you can assign block using the automatic block assignment method. For details, see *Associate Instrument Blocks Automatically (via Instrument Type)* (on page 263).

- Press F7 to open the **Domain Explorer** and then expand the hierarchy to the tag level in the **Loops** folder.
- In the **Items** pane, select and right-click tag numbers to which you want assign blocks.
- On the shortcut menu, click **Actions > Associate Instrument Blocks**.

TIPS

- If there is a large number of blocks in the data window, under **Find block**, select a column heading and then type a value. For example, if you select **Block** as a column heading and type block name **Control**, in the data window, the software locates the block whose name starts with **Control**.
 - If you want to display only specific blocks in the data window, click **Filter** and specify a filter parameter.
- In the data window, select the blocks that you want to associate and then click **OK**.
 - In the **Specify Drawing Pages for Blocks Assignment** dialog box, under **Page**, use spinners to specify pages on which you want the selected blocks to appear after loop drawing generation.

TIPS

- The **Specify Drawing Pages for Blocks Assignment** dialog box opens when the source loop number is assigned to a multi- page drawing or multi-drawing.
- In the **Loops** folder of the **Domain Explorer**, the instrument blocks that you assigned appear under the tag numbers that you selected. These blocks are indicated with the icon .

Update Block Assignments

- In the **Domain Explorer**, expand the **Loops** folder hierarchy.
- Do one of the following:
 - In the **Items** pane, select and right-click one or more loops.
 - In the tree view pane, select a specific loop and then in the **Items** pane, select and right-click tag numbers.
- On the shortcut menu, click **Actions > Update Block Assignments**.
- On the **Update Block Assignments** dialog box, select an update option as you require.

IMPORTANT When selecting the **Overwrite** option, any existing blocks associated manually with the current tag number are dissociated. For the update option descriptions and possible scenarios, see Update Block Assignments Dialog Box.

- Do one of the following:
 - To update block assignments for the displayed tag number only, click **Apply** and then click **Next** to display another tag number.

- To update block assignments for the all the tag numbers you selected in the **Domain Explorer**, click **Apply to All**.

Copy Associated Blocks to Other Instrument Tags

NOTES

- You can only copy blocks to those tags that belong to loops you assigned to the CAD generation method. In the **Domain Explorer**, the icons of such loops appear with the **C** indicator.
 - This procedure only allows you to copy blocks assigned to a specific instrument. If you want to copy all instrument blocks that all of the loop instrument contain, you need to perform a different procedure. For details, see *Copy Automatically Assigned Blocks to Tags of Other Loops* (on page 266).
1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
 2. In the tree view pane, expand the loop hierarchy to the tag level and select a source tag.
 3. In the **Items** pane, select and right-click the blocks that you want to copy to other tag numbers.
 4. On the shortcut menu, click **Actions > Copy Blocks to Tag Numbers**.
 5. In the dialog box that opens, do one of the following to find tags belonging to the loops whose generation method is CAD:
 - Click **Find** to find all the tag numbers.
 - Set search parameters and then click **Find** to display tag numbers that match the search parameters.
 6. Under **Search results**, select tag numbers to which you want to copy the blocks.
 7. Click **OK**.

TIP If any of the source blocks is associated with an instrument type, when copying the blocks, the software changes the block assignment method from automatic to manual. In the **Loops** folder of the **Domain Explorer**, under the target instrument tags, the block icons change from  to .

Copy Automatically Assigned Blocks to Tags of Other Loops

NOTES

- You can only copy blocks to those tags that belong to loops you assigned to the CAD generation method. In the **Domain Explorer**, the icons of such loops appear with the **C** indicator.
 - Using this procedure, you copy blocks at the <plant> level, that is, your target loops belong to different <units> of the current plant. If you only want to copy blocks that belong to a specific instrument or copy blocks at the <unit> level, you need to perform a different procedure. For details, see *Copy Associated Blocks to Other Instrument Tags* (on page 266).
1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
 2. In the tree view pane, right-click a loop.
 3. On the shortcut menu, click **Actions > Copy Blocks to Tags of Other Loops**.
 4. On the dialog box that opens, under **Target loop number**, select one or more loops.
- TIP** You can select all the target loops or find a specific loop. The software finds and selects the loop number as you type its name in the **Find target loop** box.

5. Click **OK** to validate the target blocks and create the copy_block.txt log file in the temporary folder path of SmartPlant Instrumentation.
6. When prompted to copy the blocks, do one of the following::
 - Click **Yes** to start copying the blocks and skip problematic loops automatically.
 - Click **No** if you only want to view the target loop validation problems in the log file.
7. In the temporary folder path of SmartPlant Instrumentation, view the copy_block.txt log file.

Assign an Instrument Block to a Drawing Page

1. In the **Domain Explorer**, expand the **Loops** folder hierarchy.
2. In the tree view pane, expand the hierarchy to the instrument level and then select an instrument tag.
3. In the **Items** pane, right-click a block.

TIPS

- You can only select one block at a time.
 - For a block shown as , you might have already specified the drawing page when assigning the block to the instrument using the following procedure: *Associate Instrument Blocks Manually* (on page 265). It is possible to specify a different page for such a block.
4. On the shortcut menu, click **Actions > Select Drawing Page**.
 5. On the **Select Drawing Page** dialog box, select a page number.
 6. Double-click the selected number or click **OK** and then repeat this procedure to assign another block to a drawing page.

Dissociate Blocks from Instruments

Use this procedure to dissociate blocks from instruments belonging to the same or different loops.

1. On the **Loop Drawings Module** window, click **Actions > Dissociate Blocks from Instruments**.
2. In the **Dissociate Blocks from Instruments** dialog box, under **Select blocks**, select one or more blocks and also select the specific instruments from which you want to dissociate the blocks.
3. Click **OK** to dissociate the blocks.

TIPS

- When dissociating blocks assigned to instruments using the automatic block assignment method, the software does not dissociate these blocks from the instrument type. You can reassign these blocks automatically whenever you require. In the **Domain Explorer**, these blocks are shown as . For details, see *Update Block Assignments* (on page 265).
- You can also dissociate instrument blocks from the **Loops** folder of the **Domain Explorer**. However, it is only possible to dissociate blocks belonging to one specific instrument. After you select an instrument in the **Loops** folder, the software displays the associated blocks in the **Items** pane. You can right-click one or more blocks and then, on the shortcut menu, click **Actions > Dissociate Instrument Blocks**.

Specify Border and Logo Blocks for Loop Drawings

NOTES

- You can only perform this procedure after you have assigned blocks to the **Border and Logo** general block types in the **Drawing Block Types** folder of the **Domain Explorer**.
 - In addition to the shipped general block types, you might have created user-defined general block types. All blocks belonging to the block types appear automatically in every CAD loop drawing that you generate.
1. In the Loop Drawings Module window, click **Actions > Default General Blocks**.
 2. In the **Default General Blocks** dialog box, select the border and logo blocks to be used in all the loop drawings.

Assign General Blocks to All Loop Drawings via Instrument Type

1. In the **Domain Explorer**, right-click the **Drawing Block Types** folder.
 2. On the shortcut menu, click **New > General Block Type** and define a block type.
 3. Right-click the block type you created and then, on the shortcut menu, click **New > Block**.
 4. Define one or more blocks as you require.
 5. In the **Loop Drawings Module** window, do one of the following:
 - On the toolbar, click .
 - Click **Actions > Block - Instrument Type Assignment**.
 6. On the **Block - Instrument Type Assignment** dialog box, in the **Type** data window column, select an instrument type.

TIP If there is a large number of instrument types in the data window, under **Find instrument type**, select a column heading and then type a value. For example, if you select **Description** as a column heading and type **Control**, in the data window, the software locates the instrument types whose description starts with **Control**.
 7. Click **Assign**.
 8. From the **Block type** list, select the user-defined general block type that you created in the **Drawing Block Types** folder.
 9. Select one or more blocks that you want to assign to the instrument type.
 10. Click **OK** and then generate the appropriate loop drawing to display the assigned blocks.
- TIP** User-defined general blocks are only shown in the **Drawing Block Types** folder of the **Domain Explorer**.

Loop Drawing Types and Drawing Properties

In the Loop Drawings module, you can specify a drawing type for every loop drawing that you want to generate.

The following drawing types are available:

- **Single-page** — A loop drawing type used to display all the drawing blocks and the title block on a single drawing page. Property updates for this drawing type take effect in both CAD loop drawings and drawings you generate by the Enhanced Report Utility.
- **Multi-page** — A loop drawing type used to display drawing blocks on more than one page, where all the pages share the same document number, description, and title block. Property updates for this drawing type only take effect in CAD loop drawings.
- **Multi-drawing** — A loop drawing type used to display blocks on more than one page, where each page has an individual document number, description, and title block. Property updates for this drawing type only take effect in CAD loop drawings.

By default, all drawings belong to the single page drawing type.

Loop drawing properties appear in the title block fields. These fields contain identifying information about the document number, page number, and file name, and also information about revisions, approvals, and references to other documents.

Properly completed identifying information ensures that each drawing is uniquely identifiable. You can maintain your loop drawing properties by keeping the revision, document reference and client / vendor approval information up to date.

Loop Drawing Types and Drawing Properties Common Tasks

The following tasks are used when defining the properties of loop drawing types and drawings.

Set the Date Format in CAD Loop Drawings

Use this procedure to set a date format in your CAD loop drawings. You set the date format when customizing your CAD loop drawing preferences on the **Preferences** dialog box.

For more information, see *Set the Date Format in CAD Loop Drawings* (on page 271).

Define a Drawing as a Multi-Drawing

Use this procedure to define a CAD drawing as a multi- drawing.

For more information, see *Define a Drawing as a Multi- Drawing* (see "Define a Drawing as a Multi-Drawing" on page 271).

Define a Drawing as a Multi-Page Drawing

Use this procedure to define a CAD drawing as a multi-page drawing.

For more information, see *Define a Drawing as a Multi-Page Drawing* (on page 272).

Modify Properties of a Single-Page Drawing

Use this procedure to modify properties of a CAD drawing to be generated as a single-page drawing.

For more information, see *Modify Properties of a Single- Page Drawing* (see "Modify Properties of a Single-Page Drawing" on page 272).

Maintain CAD Loop Drawing Revisions

Use this procedure to maintain CAD loop drawing revisions. Revisions enable you to keep track of the changes made to the CAD loop drawing during its lifetime. It is important and useful to have a chronological description of the changes, dates of change, and a list of persons who approved them. Use the following procedure to add, edit, or delete revisions.

For more information, see *Maintain CAD Loop Drawing Revisions* (on page 273).

Maintain Document References

Use this procedure to maintain document references that you include in title blocks of CAD loop drawings. You can manually add a new document reference, or you can edit or delete an existing document reference.

Title blocks can display the following document references:

Automatic references to P&ID drawings

- Manual references to loop drawings
- Automatic references to reports generated in the Wiring module

For more information, see *Maintain Document References* (on page 274).

Make CAD Drawing Approvals

Use this procedure to make client and vendor approvals for CAD loop drawings.

For more information, see *Make CAD Drawing Approvals* (on page 274).

Modify Loop Drawing Identifying Information

Use this procedure to set or modify loop drawing identifying information.

For more information, see *Modify Loop Drawing Identifying Information* (on page 274).

Update Paths for CAD Drawings and Block Files

Use this procedure to update in the SmartPlant Instrumentation database paths for existing drawings and block files. You can update paths globally for block files associated with instruments and loops, for general block files, and for generated CAD drawings.

For more information, see *Update Paths for CAD Drawings and Block Files* (on page 275).

Associating an External CAD Drawing

This option enables you to associate an external CAD drawing with one or more loops you have selected in the **Domain Explorer**. An external CAD drawing is a drawing you have created outside the SmartPlant Instrumentation environment, using SmartSketch, AutoCAD, or MicroStation. After associating an external CAD drawing, you can open it in a compatible CAD application using the **View External CAD Drawing** option of the **Domain Explorer**. External CAD drawings are only used for viewing from within SmartPlant Instrumentation. They are neither included in loop drawing generation, nor referenced in any item properties or settings.

For more information, see *Associating an External CAD Drawing* (on page 275).

View an External CAD Drawing

This option allows you to view an external CAD drawing that has been associated with one or more loops the **Domain Explorer**. An external CAD drawing is a drawing you have created outside the SmartPlant Instrumentation environment, using SmartSketch, AutoCAD, or MicroStation.

For more information, see *View an External CAD Drawing* (on page 276).

Set the Date Format in CAD Loop Drawings

1. On the **File** menu, click **Preferences**.
2. In the tree view, click **Loop Drawings > General**.
3. In the **Date format** box, enter the date format using the syntax in the following table:

Date Format	Example
mm-dd-yy	01-30-09
mm-dd-yyyy	01-30-2009
dd-mm-yy	30-30-09
dd-mm-yyyy	30-30-2009
d-mmm-yy	30-Jan-09
mmm-yy	Jan-09

TIPS

- SmartPlant Instrumentation only support date formats that are listed in this table.
- You can use one of the following characters as separators: dash (-), slash (/), space, comma, semicolon, colon, or dot.

Define a Drawing as a Multi-Drawing

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the tree view pane, right-click a loop.
3. On the shortcut menu, click **Actions > Loop Drawing List**.
4. On the **Loop Drawing List** dialog box, select the loop, and then click **Properties**.
5. On the **Loop Drawing Properties** dialog box, under **Drawing type**, click **Multi-drawing**.
6. Click **New** and then enter the values as you require.

TIPS

- You cannot change the loop number value.
- In the **Page** column, you can type any alphanumeric character that designates a specific loop drawing page.
- You can specify an individual source path for any drawing page by clicking  to the right of the selected drawing file. A source path is a path which software uses to locate the source file when generating a CAD drawing that belongs to the multi-drawing type. The path that you specify does not overwrite the path setting specified in the **Preferences** dialog box. The software saves all generated loop drawings in a designated folder that you define on the **Preferences** dialog box.

Define a Drawing as a Multi-Page Drawing

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the tree view pane, right-click a loop.
3. On the shortcut menu, click **Actions > Loop Drawing List**.
4. On the **Loop Drawing List** dialog box, select the loop, and then click **Properties**.
5. On the **Loop Drawing Properties** dialog box, under **Drawing type**, click **Multi-page**.
6. Click **New** and then enter the values as you require. Repeat this action as many times as you need.

TIPS

- You cannot change the loop number value.
- In the **Page** column, you can type any alphanumeric character that designates a specific loop drawing page.
- You can specify an individual source path for any drawing page by clicking  to the right of the selected drawing file. A source path is a path which software uses to locate the source file when generating a CAD drawing that belongs to the multi-page type. The path that you specify does not overwrite the path setting specified on the **Preferences** dialog box. The software saves all generated loop drawings in a designated folder that you define on the **Preferences** dialog box.

Modify Properties of a Single-Page Drawing

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the tree view pane, right-click a loop.
3. On the shortcut menu, click **Actions > Loop Drawing List**.
4. On the **Loop Drawing List** dialog box, select the loop number, and then click **Properties**.
TIP If the drawing belongs to the single page type, the **Single page** button under **Drawing type** is selected automatically. You can change the type of the drawing to multi-page or multi-drawing. For more information, see *Define a Drawing as a Multi-Page Drawing* (on page 272) or *Define a Drawing as a Multi-Drawing* (on page 271).
5. Modify the values as you require.

Maintain CAD Loop Drawing Revisions

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the **Items** pane, select and right-click one or more loops.
3. On the shortcut menu, click **Actions > Loop Drawing List**.
4. On the **Loop Drawing List** dialog box, click **Revisions**.
5. On the **Revisions** dialog box, select one of the revision numbering methods (use **P0**, **P1**, **P2...** for preliminary revisions or **0**, **1**, **2 / A**, **B**, **C** and so forth for normal serial revisions).

TIPS

- When you first select a revision numbering method, several options are available to you, including preliminary revisions (designated by **P0**, **P1**, **P2...**). Once you select one of the other revision methods, you can no longer return to the preliminary revision method and this option becomes disabled.
 - When working with revisions in an integrated environment, the revision numbering methods are not available for selection.
6. Click **New** to add new revision data or update the existing data as desired.

TIPS

- SmartPlant Instrumentation automatically adds a new line with the next logical character and date each time you click **New** after you select the initial method.
 - If you are working in an integrated environment in a module that supports publishing and retrieving of documents, clicking **New** opens the **Revise** dialog box for the integrated environment instead of the SmartPlant Instrumentation **Revisions** dialog box.
7. Add or edit the revision data in the appropriate data fields.

TIPS

- The **By** column contains the current user initials by default, if previously defined by the System Administrator. You can edit this value as you require.
- You can delete obsolete revisions by clicking **Delete**.
- You can also maintain revisions in batch mode.
- Each revision is represented by a numbered macro and the revisions are displayed in the order of the macro numbering, beginning with the first revision (ascending order) or the last revision (descending order). If the total number of revisions is greater than the number of lines available for display in the drawing and you want to see the latest revisions, you should set your preferences to display the revisions in descending order so that the latest revision is displayed first.

Maintain Document References

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the **Items** pane, select and right-click one or more loops.
3. On the shortcut menu, click **Actions > Loop Drawing List**.
4. On the **Loop Drawing List** dialog box, click **References**.
5. On the **Document References** dialog box, do one of the following:
 - Select a row in the data window and click **Properties** to update the existing document reference name and description of a manually inserted reference.
 - Click **New** to define properties of a new document reference.

TIPS

- You can view document references to reports generated in the Wiring module only if you have selected the **Display references to wiring reports automatically** check box in the **Loop Drawings > General** page of the **Preferences** dialog box. You cannot edit properties of these document references.
- The time of reference creation determines the order of the references in the title block. In the title block, the software displays the references in descending order, according to the time of creation. This means that the reference you create first appears in the title block at the top of the reference list.
- If an instrument in the selected loop has a reference in a P&ID drawing, this reference automatically appears on the **Document References** dialog box. It is possible, you can edit a P&ID drawing reference in the Instrument Index module.

Make CAD Drawing Approvals

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the **Items** pane, select and right-click one or more loops.
3. On the shortcut menu, click **Actions > Loop Drawing List**.
4. On the **Loop Drawing List** dialog box, click **Approvals**.
5. On the **Approvals** dialog box, set data for client and vendor approvals as you require.

Modify Loop Drawing Identifying Information

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the **Items** pane, select and right-click loops.
3. On the shortcut menu, click **Actions > Loop Drawing List**.
4. On the **Loop Drawing List** dialog box, click **Properties**.
5. Modify the data as you require.

TIPS

- Properties that you can edit are determined by the specified drawing type.
- The document number does not have to be unique.

- Properties that only apply to CAD drawings are indicated as such. Other properties, such as document number and loop drawing description, are used in both CAD loop drawings and in loop drawings generated by the Enhanced Report Utility.

Update Paths for CAD Drawings and Block Files

NOTE The paths that you specify do not overwrite the preferences settings specified for new block files and output drawings. For more information about the CAD path preferences, see Loop Drawings > CAD File Locations (Preferences).

Update Paths for Existing CAD Drawings And Block Files

1. In the **Loop Drawings Module**, click **Actions > Update Paths for Existing Drawings and Blocks**.
2. Under **Path for associated block**, do one of the following:
 - Click **Apply to drawings in the domain/project** to apply the new path to all the plants in the current domain, or to the current project in an owner operator domain.
 - Click **Apply to drawings in the current <plant> only** to apply the new path to the current plant only.
3. Type the full path to the folder where you keep your drawing blocks files or click **Browse** to navigate to the desired folder.
4. Under **Path for generated drawings**, do one of the following:
 - Click **Apply to blocks in the domain/project** to apply the new path to all the plants in the current domain, or to the current project in an owner operator domain.
 - Click **Apply to blocks in the current <plant> only** to apply the new path to the current plant only.
5. Type the full path to the folder where you keep your generated drawings or click **Browse** to navigate to the desired folder.

Associating an External CAD Drawing

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the **Items** pane, select and right-click one or more loops.
3. On the on the shortcut menu, click **Actions > Associate External CAD Drawing**.
4. On the **Select File** dialog box, select the desired SmartSketch, AutoCAD or MicroStation file.

TIP You can associate the following file types: .dwg, .dxf, .dgn, .cgm.
5. Click **Open** to associate the selected drawing.

TIPS

- If there is already an external CAD drawing associated with the current loop or loops, the new CAD drawing overwrites the existing one.
- To dissociate the external CAD drawing, on the shortcut menu, click **Actions > Dissociate External CAD Drawing**.

View an External CAD Drawing

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the **Items** pane, select and right-click one or more loops.
3. On the on the shortcut menu, click **Reports > View External CAD Drawing** to open the CAD drawing associated with the selected loop or loops.

CAD Loop Drawing Generation

SmartPlant Instrumentation allows you to generate loop drawings using external CAD applications. The same generation principles are valid for any of the listed CAD applications. The following CAD applications are supported:

- Intergraph SmartSketch
- AutoCAD
- MicroStation

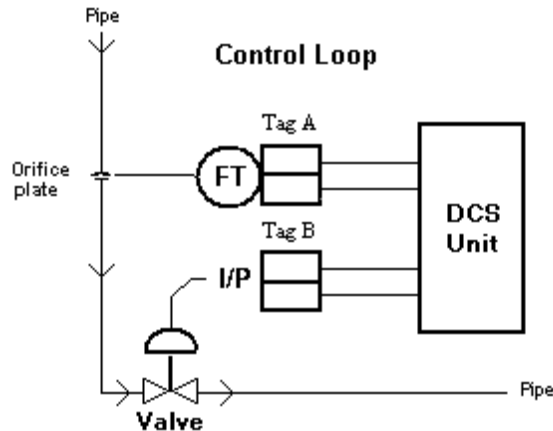
In the generation process, SmartSketch (.sym), AutoCAD (.dwg), or MicroStation (.dgn) type drawings are created from the selected loops.

Prerequisites

- Before generating loop drawings, make sure that CAD application paths are set up correctly on the **Preferences** dialog box. For details, see *CAD Settings for SmartPlant Instrumentation* (on page 278).
 - **NOTE** SmartPlant Instrumentation and the CAD drawing programs required to work with SmartPlant Instrumentation, must be installed on the same machine.
- If you use SmartSketch, prior to starting the generation process, in the **Preferences** dialog box, you must select the check box **Use macro symbol (&) to retrieve data** because in SmartSketch all macros have the '&' prefix. If you clear the check box, the software displays only macro attributes in the drawings.
- If you use AutoCAD, to ensure that the software displays macro captions, on the **Custom** tab of the **Preferences** dialog box, in the **Parameter** box, type **TagAsMacroName** and in the **Value** field, type **Y**.

Basic Principles of CAD Loop Drawing Generation

The following diagram illustrates a (schematic) flow control loop. The loop drawing includes five drawing blocks for the following components: monitoring system, control valve, each set of wiring connections to the DCS, and the DCS itself. The wiring routing and control systems are conceptually the same for both tags (but with different data), so in this example, the blocks for these items are identical.



Tag A is a monitoring system consisting of a flow transmitter and flow element.

Tag B consists of a control valve with an I/P transducer.

Loop Drawing Generation Methods

SmartPlant Instrumentation allows you to assign a loop number to one of the following generation methods: CAD, Enhanced Reports, and manual. Methods are marked by designated symbols attached to loop numbers in the **Loops** folder of the **Domain Explorer**. By default, loop numbers are not assigned to any generation method.

Loop drawing generation is traditionally done using a CAD application such as SmartSketch, AutoCAD, or MicroStation. You must specify the CAD application paths and other CAD file locations on the **Loop Drawings > CAD File Locations** page of the **Preferences** dialog box.

In addition to using a CAD method, you can also use the Enhanced Report Utility, which is supplied with SmartPlant Instrumentation. This utility enables you to generate, view, modify, and annotate loop drawings as needed. The Enhanced Report Utility has an advantage over CAD as a loop drawing generation method in that it requires no extra work in creating CAD drawing blocks. The report comes directly from the database and does not necessarily require additional modifications.

You can either assign each loop number to a specific method or apply the same method to all loop numbers when generating loop drawings in batch mode. In the **Domain Explorer**, for the intended method of generation, the software assigns an appropriate generation indicator to the loop number icons.

The available indicators are:

- **C** — Loop drawing to be generated by a CAD application (SmartSketch, AutoCAD, or MicroStation).
- **M** — Loop drawing to be generated manually, without using SmartPlant Instrumentation options. When generating loop drawings, the software always skips loops assigned to the Manual method.
- For loop drawings where the selected generation method is Enhanced Report, the following symbols are available:

- **E** — By Loop
- **ES** — By Signal
- **CL** — Custom by Loop
- **CS** — Custom by Signal

Steps in Loop Drawing Generation Using a CAD Application

SmartPlant Instrumentation enables you to generate drawings for loop numbers in your domain using an external CAD application. The main steps required to generate these loop drawings are as follows:

- **Drawing Preparations**
 - *CAD Loop Drawing Generation* (on page 276)
 - *Insert Macros in an AutoCAD Drawing Block* (on page 296)
- **Defining Blocks in SmartPlant Instrumentation**
 - *Add a Block Type* (on page 259)
 - *Blocks in SmartPlant Instrumentation* (see "CAD Drawing Blocks in SmartPlant Instrumentation" on page 253)
 - *Loop Blocks* (on page 256)
- **Setting CAD Application Parameters**
For details on preliminary Intergraph SmartSketch, AutoCAD or MicroStation settings, see:
 - *CAD Settings for SmartPlant Instrumentation* (on page 278)
 - *MicroStation and SmartPlant Instrumentation* (on page 307)
- **Customizing SmartPlant Instrumentation Macro Definitions (if required)**
 - *Customizing Macro Definitions* (on page 297)
 - *Working with User-Defined Macro Functions* (on page 299)
 - *Using External Macro Data Sources* (on page 303)
- **Making Block Associations**
 - *Associate Instrument Blocks Automatically (via Instrument Type)* (on page 263)
 - *Apply Associated Blocks to all Tag Numbers Belonging to a Specific Instrument Type* (see "Associate a Loop Block" on page 263)
 - *Associate Instrument Blocks Manually* (on page 265)
- **Performing the Generation Process**
 - *Generate CAD Loop Drawings* (on page 283)

CAD Settings for SmartPlant Instrumentation

The table below contains the CAD interface parameters together with the appropriate values for SmartSketch, MicroStation, and AutoCAD. You can modify these parameters on the **Loop Drawings > CAD File Locations** page of the **Preferences** dialog box. The **Preference** column of the table below shows the label that appears on the **Preference** dialog box.

Preference for SmartSketch	Default Value	Description
CAD folder	C:\Program\	The path to the draft.exe for SmartSketch.

Preference for SmartSketch	Default Value	Description
CAD configuration folder	C:\Program\	There is no configuration file for SmartSketch. However, the default value for the path does exist.
Temporary folder path	C:\Program Files\SmartPlant\Instrumentation\TEMP\	The path to the SmartPlant Instrumentation temporary folder where the software stores temporary logo and custom style files.
Drawing file folder	C:\Program Files\SmartPlant\Instrumentation\CAD\DWG\	The path to the location of the loop drawing that was last generated.
Drawing block folder	C:\Program Files\SmartPlant\Instrumentation\CAD\Blocks\	The default path for new drawing block files.

Preference for AutoCAD	Default Value	Description
CAD folder	C:\Acadwin\	The path to the acad.exe file for AutoCAD.
CAD configuration folder	C:\Acadwin\	The path to the AutoCAD configuration file ACAD.CFG.
CAD function folder	C:\Program Files\SmartPlant\Instrumentation\CAD\CADfunc\	The path to the AutoCAD function files (.lsp file format).
Temporary folder path	C:\Program Files\SmartPlant\Instrumentation\TEMP\	The path to the SmartPlant Instrumentation temporary folder where the software stores temporary logo and custom style files.
Drawing file folder	C:\Program Files\SmartPlant\Instrumentation\CAD\DWG\	The path to the location of the loop drawing that was last generated.
Drawing block folder	C:\Program Files\SmartPlant\Instrumentation\CAD\Blocks\	The default path for new drawing block files.

Preference for MicroStation	Default Value	Description
CAD folder	C:\Win32app\Ustation	The path to the ustation.exe file for MicroStation SE.
CAD configuration folder	C:\Win32app\Ustation\Config\	The path to the MicroStation SE configuration file MSCONFIG.CFG.

Preference for MicroStation	Default Value	Description
CAD function folder	C:\Program Files\SmartPlant\Instrumentation\CAD\CADfunc\	The path to the MicroStation SE function files (.ma file format).
Temporary folder path	C:\Program Files\SmartPlant\Instrumentation\TEMP\	The path to the SmartPlant Instrumentation temporary folder where the software stores temporary logo and custom style files.
Drawing file folder	C:\Program Files\SmartPlant\Instrumentation\CAD\DWG\	The path to the location of the loop drawing that was last generated.
Drawing block folder	C:\Program Files\SmartPlant\Instrumentation\CAD\Blocks\	The default path for new drawing block files.

NOTES

- SmartPlant Instrumentation uses the shipped LISP function plot.lsp for plotting generated drawings. If this function is not suitable for the specific current AutoCAD plotter, you can select the **Current Systems Plotter** option from the **AutoCAD Plotter Configuration** menu of AutoCAD, or modify the plate.lsp file. If you do the latter, you should be aware that the last **Enter** for the AutoCAD plot command is supplied automatically; you do not need to press **Enter** manually.
- When defining a new CAD drawing block for MicroStation, the files you select represent cell libraries. To specify a particular cell in the library, you must type its name manually in the appropriate field.
- Make sure the temporary folder path string does not exceed fifty two characters. A longer path prevents MicroStation from generating or displaying drawings or cells from SmartPlant Instrumentation.
- SmartPlant Instrumentation and the CAD drawing programs required to work with SmartPlant Instrumentation, must be installed on the same machine.

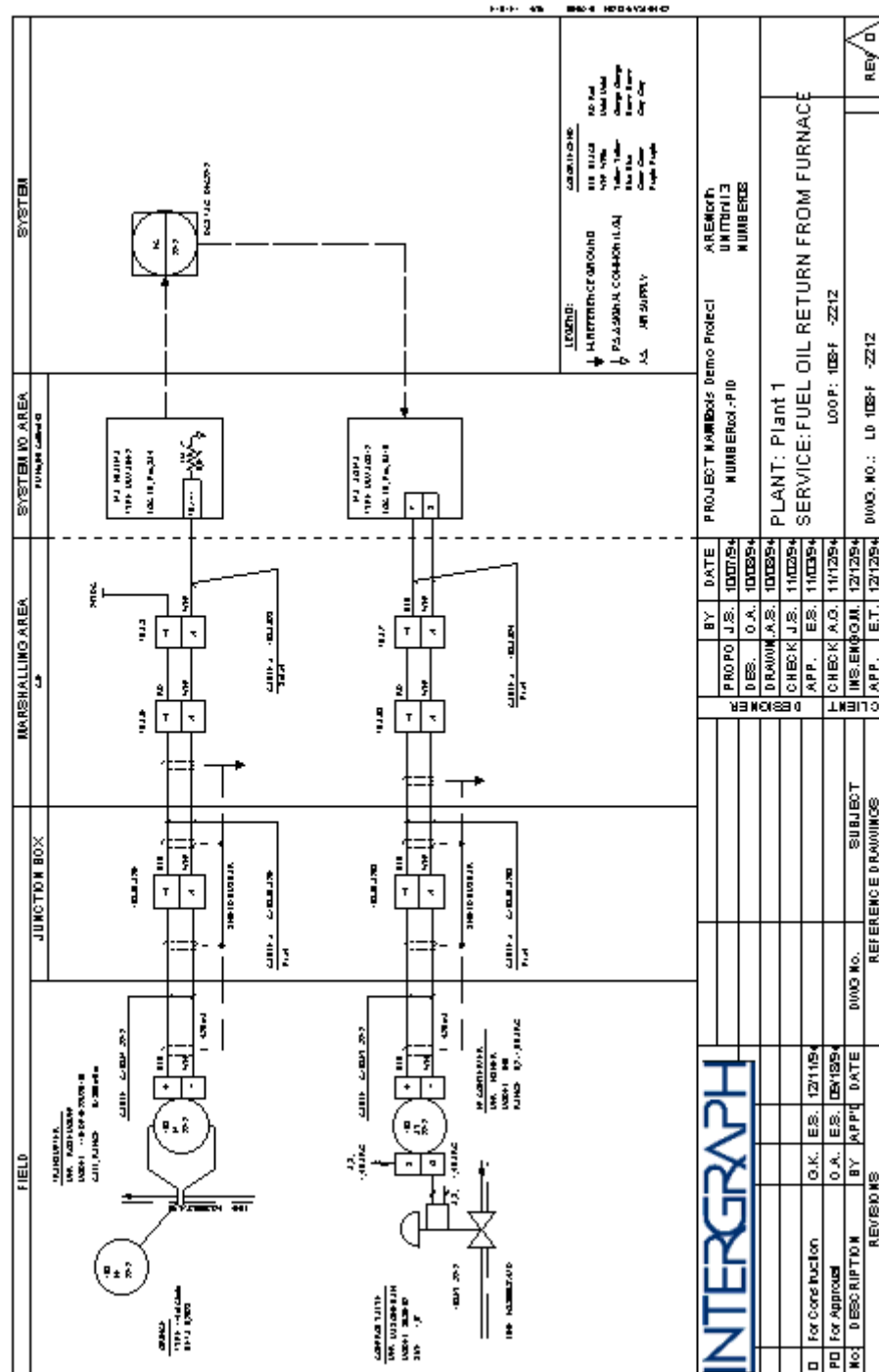
See Also

Loop Drawing Generation Methods (on page 277)

CAD Loop Drawing Generation (on page 276)

CAD Loop Drawing Generation Common Tasks (on page 282)

Sample Loop Drawing (After Generation)



CAD Loop Drawing Generation Common Tasks

The following tasks are used when you generate CAD loop drawings.

Apply Loop Drawing Generation Method

You can specify the desired method for one or more selected loops in the **Domain Explorer**. You can also specify in the **Preferences** dialog box which generation method to use by default and whether to display each signal on a separate drawing sheet when you select **By Signal** or **Custom by Signal** as the generation method.

For more information, see *Apply Loop Drawing Generation Method* (on page 283).

Generate CAD Loop Drawings

You use this procedure to generate CAD loop drawings.

For more information, see *Generate CAD Loop Drawings* (on page 283).

Show the Latest Generated CAD Drawing

This option enables you to display the latest loop drawing that was generated in SmartPlant Instrumentation using an external CAD application. The drawing opens in the CAD application you set as default on the **Preferences** dialog box.

For more information, see *Show the Latest Generated CAD Drawing* (on page 285).

Display CAD Loop Drawing Generation History

You can keep track of all the macro data changes in your CAD drawings. You choose this option when generating a loop using an external CAD engine. You can access the loop generation history from the print preview of loop drawings and the relevant reports.

The loop generation history feature compares the retrieval values of macros from different generations. Therefore, the history data is available when you have already generated at least one loop drawing for a selected loop.

For more information, see *Display CAD Loop Drawing Generation History* (on page 285).

Delete CAD Loop Drawing Generation History

You use this procedure to improve performance when comparing generated data values in CAD loop drawings, by deleting the history of obsolete CAD loop drawing generations. Before deleting the values of previous CAD loop drawing generations, you select the required loops, and then set the period within which you want to delete the selected history records.

For more information, see *Delete CAD Loop Drawing Generation History* (on page 286).

Clear Previous Macro Result Records

Use this procedure to clear macro result records of generated loop drawings. The software saves the macro results in the database for the next time you want to generate a loop drawing for the same loop. The saved data accumulates and at a certain stage might affect the generation performance.

For more information, see *Clear Previous Macro Result Records* (on page 286).

Apply Loop Drawing Generation Method

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. Do one of the following:
 - In the tree view pane, right-click a target loop.
 - In the **Items** pane, select and right-click several loops.
3. On the shortcut menu, click **Apply Generation Method** and then select a generation method.

TIPS

- In the **Domain Explorer**, the icons of the loop numbers to which you applied a generation method appear with method indicators. For method indicator descriptions, see *Loop Drawing Generation Methods* (on page 277).
- To clear the current loop generation method for one or more loops, on the shortcut menu, click **Clear Generation Method**.

Generate CAD Loop Drawings

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the **Items** pane, select and right-click one or more loop numbers.

TIP If among the loops that you selected there are loops assigned to the Manual method (symbol **M** in the **Explorer**), the software skips these loops when generating drawings in batch mode.
3. On the shortcut menu, click **Reports > Generate Loop Drawings**.
4. On the **Generate Loop Drawings** dialog box, from the **Generation method** list, select **CAD**.
5. Under **Automatic save options**, select **Save drawing automatically** and do one of the following:
 - In the **Output drawing path** data field, type the exact folder path to which you want to save the generated drawing.
 - Click **Browse** to navigate to the desired path for saving the file.
6. Click **CAD Options**, and then, from **Save file in format** list, specify the CAD file format in which to save the generated drawings (if your CAD application supports more than one file format).
7. Select the following check boxes:
 - **Send to plotter or printer** — Sends the generated drawing to the current CAD plotter.
 - **Use macro functions** — Applies macro functions associated with the macros.
 - **Save output data to database** — Enables you to compare drawing data of the current generation to data of a subsequent generation made for the same loop.
8. Under **Generation criteria**, do one of the following:
 - Click **Per loop** to generate drawings for all the loops you selected.
 - Click **Per page** and then type the desired page number to generate only a specific drawing page for the drawings you selected.
 - Click **Per block** to select a block from the list and generate drawings only for the loops that to which you assigned this block.

TIP The **Per block** option is useful when you select several loops in the **Items** pane of the **Domain Explorer**. If your selection includes loops that do not contain the block that you selected, the software does not generate loop drawings for those loops. For the loops that do contain the selected block, the software includes all of the blocks when generating loop drawings. For example, if you selected five loops and each of them has the block that you selected and also has three other different blocks, the software includes all of these blocks in the generated loop drawings. The blocks displayed in the lists are blocks associated with instruments using the manual block assignment method. In the **Loops** folder of the **Domain Explorer**, these blocks are shown as are shown with the  icon.

9. Under **Create report about**, select which reports you want to generate:
 - **Null data** — Displays macro strings in the drawing for which SmartPlant Instrumentation data is unavailable.
 - **Invalid macros** — Displays macro strings in the drawing which are not defined in SmartPlant Instrumentation.

TIP You can select to generate both reports if needed.

10. Under **Mark drawing using**, select the options for which you want to display a text string, specified in the adjacent text box:
 - **String for null data** — Allows you to type a string that indicates in the drawing those macros for which SmartPlant Instrumentation data is unavailable.
 - **String for invalid macros** — Allows you to type a string that indicates in the drawing those macros which are not defined in SmartPlant Instrumentation.
11. If you need to use macros from an external source during the current loop drawing generation, under **External data**, select the **Use external macro source** check box, and then click **Define**.
12. Click **OK** to start the loop drawing generation process in your CAD application.

TIPS

- If your CAD application is SmartSketch, by default, all drawings that you generate have the A3 Wide print sheet size. This print sheet size is defined in the template file pid.igr, which comes shipped with SmartPlant Instrumentation. This file appears in the CADFunc folder, for which you specified the path setting on the **Preferences** dialog box, **Loop Drawings > CAD File Locations** page. If you want to generate loop drawings using a different print sheet size, you must open the pid.igr file in SmartSketch and modify the sheet settings as you require. Note, however, that changes that you make in the pid.igr file affect both loop drawings and hook-up drawings you generate in SmartSketch. If you want the print sheet size of loop drawings not to affect the print sheet size of hook-up drawings, duplicate the CADFunc folder with all its content and define a different path setting preference for hook-up drawings on the **Hook-Ups > CAD File Locations** page.
- SmartPlant Instrumentation and the CAD drawing programs required to work with SmartPlant Instrumentation, must be installed on the same machine.

Show the Latest Generated CAD Drawing

This option enables you to display the latest loop drawing that was generated in SmartPlant Instrumentation using an external CAD application. The drawing opens in the CAD application you set as default on the **Preferences** dialog box.

- In the Loop Drawings module, click **Actions > Show Latest Generated Drawing**.

NOTE You must have checked the **Save drawings automatically** check box and entered the path for the **Output drawing folder**, under **Preferences > Loop Drawings > CAD File Locations**, for this procedure to work.

Display CAD Loop Drawing Generation History

1. On the **Generate Loop Drawings** dialog box, in the **CAD Options** tab, select the **Save output data to database** check box.
2. On the **CAD History** tab, select the **Compare outputs** check box if this is not the first generation to make the additional options available.
3. Do one of the following:
 - Click **Set date range** and then, in the **From** and **To** boxes, specify the history date range you need for the comparison.
 - Click **Select specific generation**, and then, from the data window, select a specific generation to which you want to compare the current generation.
4. Under **History data**, select one of the following check boxes or both check boxes:
 - Select **Display in the drawing** to display history data in the generated drawing.
 - Select **Display in the report** to display the **Data Changes Report** before proceeding with the loop generation. The report includes the tag number, block name, macro name and description, old value and the new value.
5. Do one of the following:
 - Click **Show data from previous generation** to display the old history value from the previous generation.
 - Click **Mark with user- define value** and type a new value in the box below. This marks the drawing or the report with the new value instead of the old history value from the previous generation.

TIPS

- In the generated drawing or report, the software displays the old value in parentheses.
- If you selected **Display in the report**, the software displays the **Data Changes Report** before proceeding with the loop generation. After printing or viewing the report, click **OK** to return to the **Generate Loop Drawings** dialog box.

6. Click **OK** to start the generation process.

NOTES

- When comparing a loop drawing that you have just generated with an older one, you have to set the comparison date range to include both generations. If that date includes more than one loop drawing, select the exact loop drawing to which you want to compare the currently generated one.
- The results show a comparison of the current data in the loop drawing with the data stored in the history for the specified date range or generation.

Delete CAD Loop Drawing Generation History

1. In the **Domain Explorer**, right-click loops that you select, and then, on the shortcut menu, click **Actions > Delete Generation History Records**.
2. In the **From** and **To** boxes, specify the date range of the generation history records that you want to delete.
3. Click **Show Records**.
4. Under **Select records for deletion**, select history records that you want to delete.
TIP Select the **Select all** check box to be able to delete all the displayed records at once.
5. Click **Delete**.

See Also

CAD Loop Drawing Generation Common Tasks (on page 282)

Clear Previous Macro Result Records

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the **Items** pane, select and right-click loops whose temporary generation data table you want to clear.
3. On the shortcut menu, click **Actions > Clear Temporary Data**.

Loop Drawing Reports

The following reports are associated with loop drawings that you can generate in SmartPlant Instrumentation:

- **CAD Loop Drawing Generation Errors** — Available on the shortcut menu in the **Domain Explorer**. This report lists a summary of errors that occurred during the last loop drawing generation using your CAD application. Data that appears in this report depends on specific loop numbers that you select. For details, see *Generate a Report of CAD Loop Drawing Generation Errors* (on page 287).
- **Macros** — Available in the Loop Drawings module, on the **Reports** menu of the main window. Lists all the SmartPlant Instrumentation macros. This report provides you with information about every macro name, description, database name, and function description (if it exists). This report displays macros for all loop numbers existing in your database. For details, see *Generate a Macro Report* (on page 295).
- **User-Defined Macro Functions** — Available in the Loop Drawings module, on the **Reports** menu of the main window. Lists all the macro functions you created on the basis of SmartPlant Instrumentation standard macro functions. The macro function affects the value which the macro returns after generation. For example, you can specify a user-defined function that removes all the spaces in the retrieved values and displays the remaining characters as upper case. This report displays user-defined macro functions for all loop numbers in your database.

Generate a Report of CAD Loop Drawing Generation Errors

Use this procedure to create a CAD loop drawing generation error report for loop numbers for which you generated CAD loop drawings.

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
2. In the tree view pane, right-click a loop for which you want to generate the CAD loop generation error report.
3. On the shortcut menu, click **Reports > CAD Loop Drawing Generation Errors**.

Working with Macros in SmartPlant Instrumentation

A macro is a text string that is recognized by SmartPlant Instrumentation and is used to extract data from a given item such as a loop, instrument, panel, strip, and so forth. Each macro corresponds to specific information available in SmartPlant Instrumentation database, for example, the color of a wire. SmartPlant Instrumentation provides you with easy ways to search for the macros that need to be placed on a drawing block or a symbol. The software also comes with a standard set of macros that you can customize according to your needs.

The type of data that the software can retrieve depends on whether you are working with loop drawings or hook-up drawings. For loop drawings, you can retrieve data for the following item types:

- Domain definitions
- Plant hierarchy items
- Documents
- Revision information
- Loop data and instrument tag numbers
- Wiring data (including connection information)
- Process data
- Specifications data
- Loop drawing data
- Hook-up drawing data

For hook-up drawings, you can retrieve data for the following item types:

- Domain definitions
- Plant hierarchy items
- Documents
- Revision information
- Instrument tag numbers
- Process data
- Hook-up drawing data

All of the standard SmartPlant Instrumentation macros are displayed on the **Macro Definitions** dialog box.

You can use SmartPlant Instrumentation macros in:

- Enhanced reports — You can add macros to the various symbols used to represent items that appear in enhanced reports using the Symbol Editor, or you can add macros directly in an enhanced report after generation. For further details, see *Using Macros: An Overview* in the *Enhanced Report Utility User's Guide*.

- CAD drawings — To display data in CAD drawings, you must add them to the blocks that are used in generating the drawings. For further details, see *Using Macros in CAD Blocks* (on page 295).

NOTE Macros in a CAD application, that you want to solve with a general signal in SmartPlant Instrumentation, must have the general signal name (as defined in the SmartPlant Instrumentation **Local Signal** dialog box) added as a prefix to the CAD application macro. For example, for the CAD macro **PNL_NAME.1.4**, to be recognized by SmartPlant Instrumentation and solved with the general signal called **GENERSIG1**, you add the general signal name to the CAD macro as a prefix in the CAD application: **GENERSIG1.PNL_NAME.1.4**.

See Also

Working with User-Defined Macro Functions (on page 299)

Standard Functions (on page 300)

Customizing Macro Definitions (on page 297)

SmartPlant Instrumentation Macro Structure

You select the SmartPlant Instrumentation macros that you want the software to recognize in hook-up drawings or in loop drawings by defining custom macro names on the **Macro Definitions** dialog box for the appropriate drawing type. To see the entire list of SmartPlant Instrumentation macros, you can generate a macro report in the Loop Drawings module. For details, see *Generate a Macro Report* (on page 295).

Certain SmartPlant Instrumentation macros that include a group level have the following structure:

[Macro_name.x]

The letter 'x' designates the group level. The group level depends on the number of macros in a group. The software creates a group when several macros of the same macro type are associated with a tag number. For example, when associating several tag categories with a tag number, the system combines all the TAG_CAT macros into a TAG_CAT.x group so that you set a separate level for each macro: TAG_CAT.1, TAG_CAT.2, TAG_CAT.3, and so forth.

For CAD drawings only, if desired, you can associate each macro with a macro function, which determines how the software performs string manipulation on the target data. For example, a macro function can retrieve the entire target data string or just part of the data string and display it on the generated drawing. Associating a macro function with a macro is optional. For details, see *Working with User-Defined Macro Functions* (on page 299).

The structure of a macro including a user-defined macro function is as follows:

[Macro_name.F1]

where F1 is the macro function abbreviation. If you set a macro function as a default, the software automatically adds the abbreviation of the function to the macro in the actual CAD drawings (SmartSketch, AutoCAD and MicroStation). If you do not set a macro function as default, you need to add the macro function abbreviation manually on the drawing.

NOTES

- The separator in a macro string is a period (.), therefore within a macro name segment, periods are not allowed.
- Some of the wiring macros have a different structure. For details, see *Wiring Module Macros Conventions* (see "Wiring Module Macros - Conventions" on page 293).

Loop Block Macro Syntax

There are nine different permutations for the syntax used to construct macros for use with CAD loop drawings in SmartPlant Instrumentation, four macros for use with loops, loop blocks, and so forth, and five for use with wiring. The different segments that make up the macros are:

- **Macro_Name** — the macro name, for example **Tag_Num** or **Cbl_Num**
- **IT** — instrument type, for example **FY**
- **F1** — user defined function name, set in the **User Defined Macro Function Properties** dialog box, for example **Trim**
- **X** — the group level (wiring macros), for example **2**
- **Y** — the group sequence (wiring macros), for example **3**
- **Suffix** — The tag suffix as used in the tag naming convention.

NOTE Each segment must be separated by a period (.) from the next segment

The four syntax permutations for loop block macros are:

- Macro_Name.IT
- Macro_Name.F1.IT
- Macro_Name.IT.Suffix
- Macro_Name.F1.IT.Suffix

The five syntax permutations for wiring macros are:

- Macro_Name.F1.X.Y
- Macro_Name.X.Y.IT
- Macro_Name.F1.X.Y.IT
- Macro_Name.X.Y.IT
- Macro_Name.F1.X.Y.IT.Suffix

Title Block Macro List

You specify macros in your title block by adding SmartText and defining the appropriate macro in the user properties. The following list summarizes the macros available for use in enhanced report title blocks (where a macro name appears in parentheses, the main name is the macro alias while the name in parentheses is the actual database name).

General

Name	Description
plant_name or plnt_name	Plant name
area_name	Area name
sub_area_name	Sub-area name
unit_name	Unit name
unit_num	Unit number
unit_note	Note added for unit
city	City where plant is located
location	Location of plant

Name	Description
owner (owner_name)	Owner name
domain_name (proj_name)	Domain name
domain_desc (proj_desc)	Domain description
eng_proj_name <i>or</i> proj_name (engineer_name)	Project name
eng_proj_desc <i>or</i> proj_desc (engineer_desc)	Project description
date	Date of drawing generation
time	Time of drawing generation

Loop

Name	Value	Description
loop_name		Loop name
loop_serv		Loop service
loop_note		Loop note
ref_name	1..5	Loop drawing reference name (for revisions 1 to 5)
ref_desc	1..5	Loop drawing reference description (for revisions 1 to 5)

Strip & Panel

Name	Description
panel_name	Panel name
panel_note	Panel note
strip_name	Strip name

Segment & Profibus

Name	Description
group_name	Segment name
wire_group_note	Segment note

Telecom Line

Name	Description
line_name (tc_line_number_name)	Telecom line name

Telecom Class

Name	Description
class_name (tc_net_cls_name)	Telecom class name

Telecom Speaker

Name	Description
cmpnt_name	Telecom speaker name
cmpnt_note	Telecom speaker note

Cable

Name	Description
cable_name (cable_num)	Cable name
cable_note	Cable note

Cable Harness

Name	Description
harness_name (cable_harness_name)	Cable harness name

Location

Name	Description
location_name (panel_loc_name)	Panel location

Rack

Name	Description
rack_name (cabinet_rack_name)	Cabinet rack name

Hook-Up

Name	Description
hu_name	Hook-Up name
gen_output_file	Hook-Up output file name
hu_udf_co1	Hook-Up user defined fields
hu_type_desc	Hook-Up description

Drawing

Name	Value	Description
dwg_name		Drawing name
dwg_desc		Drawing description
dwg_num		Drawing number
cl_ck_by or cl_chk_by		Client checked by
cl_ck_date or cl_chk_date		Client checked date
cl_engr_by		Client engineered by
cl_engr_date		Client engineered date
cl_appr_by		Client approved by
cl_appr_date		Client approved date
ven_appr_by		Vendor approved by
ven_appr_date		Vendor approved date
ven_chk_by or ven_ck_by		Vendor checked by
ven_ck_date or ven_chk_date		Vendor checked date
ven_drwn_by		Vendor drawn by
ven_drwn_date		Vendor drawn date
ven_dsgn_by		Vendor design by
ven_dsgn_date		Vendor design date
ven_prop_by		Vendor proposed by
ven_prop_date		Vendor proposed date
loop_func (loop_func_name)		Loop function
dwg_udf_c01... - ...dwg_udf_c102		Drawing user- defined character fields
loop_udf_c01... - ...loop_udf_c50		Loop user-defined character fields
curr_rev_name (current revision number)	1	Current revision name
pid_name		P&ID drawing name
Sheetno		Current sheet number
Sheetttotal		Total number of sheets
Sheetcount		<sheetno> of <sheetttotal>

Revisions

These macros have the structure [Macro_name.x.f1] where x is the revision number (max.value =5), and f1 is the abbreviation for a user-defined function (optional).

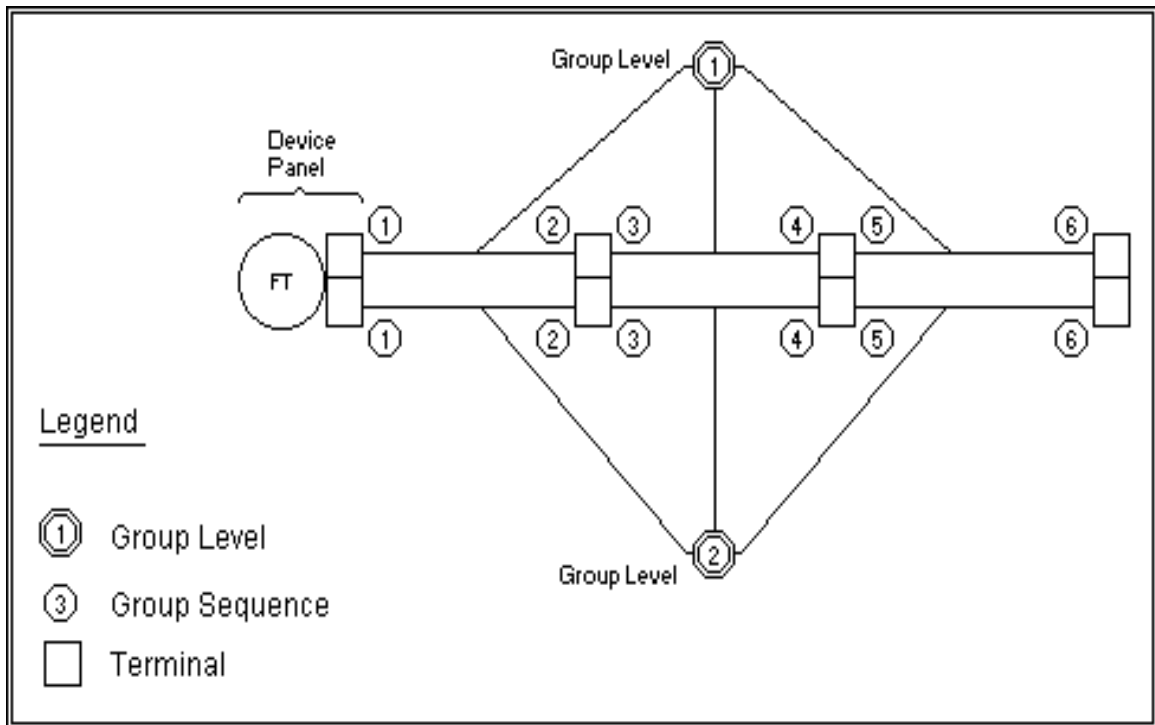
Name	Value	Description
dwg_rev_num <i>or</i> rev_no	1..5	Drawing revision number (for revisions 1 to 5)
dwg_rev_desc <i>or</i> rev_desc	1..5	Drawing revision description (for revisions 1 to 5)
dwg_rev_date <i>or</i> rev_date	1..5	Drawing revision date (for revisions 1 to 5)
curr_rev_num		Last active revision
rev_create_by <i>or</i> rev_drawn_by	1..5	Revision created by (for revisions 1 to 5)
rev_sign	1..5	Revision signed by (for revisions 1 to 5)
rev_chk_by	1..5	Revision checked by (for revisions 1 to 5)
rev_appr_by	1..5	Revision approved by (for revisions 1 to 5)
rev_udf_c01... - ...rev_udf_c20		Revision user-defined character fields

Wiring Module Macros - Conventions

To retrieve wiring information, SmartPlant Instrumentation uses macros that are identified by their macro name, together with appropriate parameters needed to retrieve specific tag data.

When you create a tag number and a matching device panel, the software creates a **group** that includes all the wires associated with that specific tag. Each wire can belong to one group only. Once you connect a cable with the device panel, the software sorts the wires into different **group levels**. Every connection of a wire to a terminal is assigned a **group sequence** that maps and numbers the connections starting with the value '1' that is usually on the device panel side.

The following diagram is an example that explains the meaning of wire groups.



Before you can start to build typical wiring drawing blocks, you need to know:

- How to identify typical wiring routing.
- The number of routing levels, usually determined by the number of wires connected to the device-panel, but this may be different if, for example, a power supply is involved.
- The group sequence that each wire and terminal connection has in the current typical routing.

Syntax

The syntax used for wiring macros is as follows:

[Macro_name.x.y]

where the letter 'x' designates the group level and the letter 'y' designates the group sequence.

Examples

The following examples show how the wiring macro syntax is used to retrieve specific data for a terminal and for a wire:

- To retrieve the number of the terminal at Group Level 1 and Group Sequence 3, the required macro is TERM_NUM.1.3.
- To retrieve the color of the wire at Group Level 2 and Group Sequence 2, the required macro is W_CLR.2.2.

NOTES

- When retrieving data for a wire, you may use the group sequence corresponding to either of the wire ends, so for example, to extract the wire name for the wire at Group Level 2 whose ends have sequence numbers 3 and 4, you may use either macro W_TAG.2.3 or W_TAG.2.4.
- Control system macros do not require the 'y' (group sequence) parameter.

- If you generate a loop drawing with tag numbers that have incomplete wiring, the macros will **not** be able to retrieve all the needed wiring information since propagation is bi-directional: from the field device and from the Control System.
- The software does not order wire sequence numbers sequentially until the propagation process finds signal continuity between the field device and the Control System.
- Special macros exist to extract data related to an overall shield. The macro for a wire that is used as an overall shield has the format [OSHW_TAG.x.y.] and the macro for a terminal to which an overall shield is connected has the format [OSHT_NUM.x.y.]. For these macros to function properly, the following conditions must be met:
 - The polarity of the wire must be designated as **Shield**.
 - The level and sequence of at least one of the wires in the cable must be defined.
 - For CAD drawings only, if desired, you can associate each macro with a macro function, which determines how the software performs string manipulation on the target data. For example, a macro function can retrieve the entire target data string or just part of the data string and display it on the generated drawing. Associating a macro function with a macro is optional.
- The general format of a wiring macro that includes a user-defined macro function is [Macro_name.F1.x.y].

See Also

Working with User-Defined Macro Functions (on page 299)

Generate a Macro Report

1. Start the Hook-Ups or Loop Drawings module.
2. Click **Reports > Macros**.
3. On the **Macro Report Print Request** dialog box, do one of the following:
 - Click **No grouping** if you want to generate a report in which all the macros are displayed in alphabetical order.
 - Click **Group by item type** if you want to generate a report in which the macros are grouped alphabetically by item type.
4. Click **OK** to generate the report.

TIP When you group the macros by item type, the software organizes the report pages according to the module titles.

Using Macros in CAD Blocks

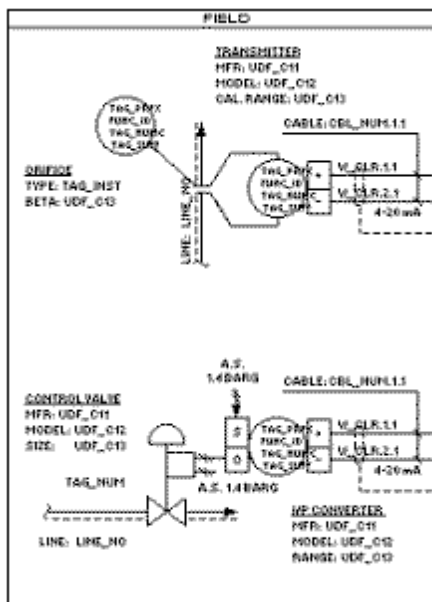
A macro enables you to perform a number of automated actions in a predefined order. When using a CAD application, macros are used to retrieve data from the database to a predefined location on the drawing layer. The macros are assigned per block and the text that they retrieve is eventually displayed as part of the block.

You can assign one or more macros to each block. Some macros are used for retrieving data for instruments with the same function. For example, flow transmitters, flow elements, and I/P transducers have a common function. Therefore the same macros are used to retrieve the data for each of these instruments. The macros required for each of these instruments are the tag prefix (TAG_PRFX), instrument type (FUNC_ID), tag number (TAG_NUMC), and tag suffix (TAG_SUFF).

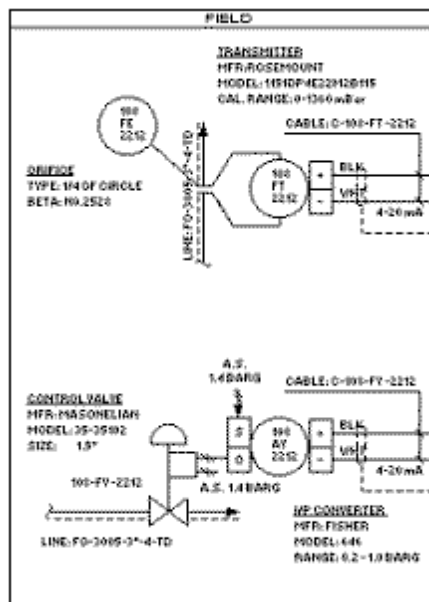
The diagrams below display a loop drawing that was generated along with its related macros. This example illustrates how the same macros can be used to extract the same or different data for different tag numbers – the Tag Prefix and Tag Number are identical for each of the three instruments. However, the instrument type is different, although represented by the same macro.

NOTE For the instruments in this example, the tag suffix was not used.

Loop Drawing with Macros



Generated Loop Drawing



If you have used the standard drawing blocks supplied with SmartPlant Instrumentation, or if you created your own drawing blocks with the predefined macros, SmartPlant Instrumentation recognizes these macros when generating the loop drawings, and you do not need to make any modifications to them. However, you may want to modify macros in the following circumstances:

- If your drawings include macro names other than the predefined ones, you can customize the default macro definitions as required. For details, see *Customizing Macro Definitions* (on page 297).
- If you want to modify certain display formatting of the macros, you can specify user-defined macro functions. For details, see *Working with User-Defined Macro Functions* (on page 299).
- If you want to use an external data source other than the SmartPlant Instrumentation database for displaying information defined by a macro, you can connect to the appropriate database and use the macros defined in that database. For details, see *Using External Macro Data Sources* (on page 303).

NOTE Macros in a CAD application, that you want to solve with a general signal in SmartPlant Instrumentation, must have the general signal name (as defined in the SmartPlant Instrumentation **Local Signal** dialog box) added as a prefix to the CAD application macro. For example, for the CAD macro **PNL_NAME.1.4**, to be recognized by SmartPlant Instrumentation and solved with the general signal called **GENERSIG1**, you add the general signal name to the CAD macro as a prefix in the CAD application: **GENERSIG1.PNL_NAME.1.4**.

See Also

Steps in Loop Drawing Generation Using a CAD Application (on page 278)

Command line prompt	Action
ATTDEF	Press Enter
Attribute modes	Press Enter
Attribute Tag:	Type in [macro name] and press Enter
Attribute prompt:	Press Enter .
Default attribute value	Type in [macro name] and press Enter
Justify/Style/<Start point>:	Position cursor and click once to define the bottom left position of the string.
Height:	Position cursor and click once to determine the height of the macro text. (The height of the macro text is also dependent on the current font size.)
Rotation angle <0>:	Type in a number to represent the angle of rotation of the macro text from the horizontal (the default is 0).

TIP You can have AutoCAD automatically enter most of the above parameters by using the ATT AutoLISP application. This application is automatically installed if you select AutoCAD as your default CAD interface. By using the ATT application you only need to enter the macro name once, and enter the start point, height and rotation parameters.

Customizing Macro Definitions

The database contains a number of default macros predefined for convenience of use and uniformity. If needed, for any default macro, you can assign one or more custom macro names or change the macro description on the **Macro Definitions** dialog box. You can also associate a macro function with a particular macro name.

NOTES

- Other users working in the same domain or in the same project (when the domain type is **Owner operator**) can use your customized macro definitions.
- A macro name can contain a maximum of 30 characters.
- If you want to assign a macro function to a macro directly on a drawing, you can add a function abbreviation of up to 4 characters to the macro. The total number of allowed characters, including separators (periods) is 50.
- To see the entire list of SmartPlant Instrumentation macros, you can generate a macro report for the type of drawings you are working with (hook-ups or loop drawings). For more details, see *Generate a Macro Report* (on page 295).
- Plug and Socket macros, for example PIN_NAME, are not supported in CAD generation.

See Also

Create a New Macro Definition (see "Create a New Hook-Up Macro Definition" on page 298)

Delete a Macro Definition (see "Delete a Hook-Up Macro Definition" on page 298)

Create a New Hook-Up Macro Definition

1. From the Hook-Ups or Loop Drawings module, do one of the following:
 - Click **Tables > Macro Definitions**.
 - Click .
2. Select the desired macro by choosing the item type and property, and where applicable, the relation between a specified item type and its parent. To do this, make a selection from each of the lists at the top of the dialog box.
3. In the **Macro definition details** data window, select the desired macro definition.

TIP If there are a large number of macro definitions in the **Macro definition details** data window, under **Find macro definition**, select a column heading and then type a value. For example, if you select **Function** as a column heading and type a macro function value, in the data window, the software locates the macro definition to which that macro function belongs.
4. To create a customized macro definition, do one or more of the following:
 - a. Under **User-Defined Macro Name**, type the desired user name for your macro.

IMPORTANT The **User-Defined Macro Name** field must contain a name, it cannot be left empty. If a name is missing, the macro will not function and data could be lost.
 - b. Click in the **Description** field and change the description as desired.
 - c. Under **Macro Function**, select a macro function to modify the appearance of the retrieved data.
 - d. To add another customized macro definition, click **New** and repeat the previous step.
 - e. Click **Apply** to accept your customized macro definitions.

See Also

Using Macros in CAD Blocks (on page 295)

Delete a Hook-Up Macro Definition

1. From the Hook-Ups or Loop Drawings module, do one of the following:
 - Click **Tables > Macro Definitions**.
 - Click .
2. Select the desired macro by choosing the item type and property, and where applicable, the relation between a specified item type and its parent. To do this, make a selection from each of the lists at the top of the dialog box.
3. In the **Macro definition details** data window, select the row containing the macro definition that you want to delete.

TIP If there are a large number of macro definitions in the **Macro definition details** data window, under **Find macro definition**, select a column heading and then type a value. For example, if you select **Function** as a column heading and type a macro function value, in the data window, the software locates the macro definition to which that macro function belongs.
4. Click **Delete**.

See Also

Using Macros in CAD Blocks (on page 295)

Working with User-Defined Macro Functions

You can perform string manipulation on retrieved data by assigning user-defined functions to the macro that retrieved the data. You create user-defined macro functions on the basis of SmartPlant Instrumentation standard macro functions.

The macro function affects the display formatting of the data which the macro returns after generation. For example, you can specify a user-defined function that removes all the spaces in the retrieved values and displays the remaining characters as upper case.

You can apply functions to macros directly on the CAD drawing or in the database.

Priority for Macro Functions

The order of priority for macro functions is as follows:

1. CAD drawing level — A macro function that you specify at the CAD drawing level has the highest priority for execution. In this case, you type the macro function abbreviation directly in the generated drawing in your CAD application. At this level, you can define one function per macro.
2. Database level — The next highest level of priority for execution is when you define a function with a specific macro.
3. Database level — The lowest priority for execution is given when you define a default function. To learn how to set a macro function as default, see *Create a User-Defined Macro Function* (on page 302).

Macro Function Syntax

If you want to apply the highest macro function priority to the data retrieval, you can associate a function with a macro directly on the CAD drawing block using the function abbreviation. The function abbreviation is a string with maximum of 4 characters, the first of which must be a letter. The general structure of a macro with a function is:

[Macro_name.F1]

where F1 is the macro function abbreviation.

The general format of a wiring macro that includes a user-defined macro function is:

[Macro_name.F1.x.y]

Example

The following is an example of a macro with a function:

W_CLR.1.1.RS

In this example, the macro name is W_CLR.1.1 and the macro function is RS (remove spaces).

NOTES

- Use a period (.) as the separator to separate the individual macro segments.
- The software uses the macro function parameters.
- When using a macro function at the database level, you should not define that macro function on the drawing block or in the generated drawing. For the software to recognize macro functions defined at the database level, on the **Preferences** dialog box, under **Loop Drawings > General**, select the **Use macro functions** check box. If you set a macro function as default, the software automatically adds the abbreviation of the function to the macro in the actual CAD drawings, and you do not need to add it with every block and macro. If you do not set a macro function as default, you need to add the macro function abbreviation manually on the drawing.

- Macros in a CAD application, that you want to solve with a general signal in SmartPlant Instrumentation, must have the general signal name (as defined in the SmartPlant Instrumentation **Local Signal** dialog box) added as a prefix to the CAD application macro. For example, for the CAD macro **PNL_NAME.1.4**, to be recognized by SmartPlant Instrumentation and solved with the general signal called **GENERSIG1**, you add the general signal name to the CAD macro as a prefix in the CAD application:
GENERSIG1.PNL_NAME.1.4.

See Also

Customizing Macro Definitions (on page 297)

Standard Functions

A standard macro function is a function that is supplied with the software and contains a set of predefined commands. You can associate a standard function with SmartPlant Instrumentation macros or specify a combination of standard functions to create a user-defined macro function.

SmartPlant Instrumentation has seven standard macro functions. The following table shows how each of the standard functions modifies the macro string.

Function	Description	Resulting String	Comments
None	Original string	FT – 100 a[space][space]	
Trim	Trims trailing spaces	FT – 100 a	Does not affect internal spaces
Upper	Changes characters to upper case	FT – 100 A[space][space]	
Lower	Changes characters to lower case	ft – 100 a[space][space]	
Substr (3,5)	Returns a substring of the original string	- 10	In this example, the substring starts from 3rd character and extends to 5 characters including the starting character
RemSpace (1)	Clears internal spaces when exceeding the maximum number of allowed spaces.	FT - 100 a[space][space]	In this example, where internal spaces exist, a maximum of 1 space is allowed. Does not affect trailing spaces.
RemSpace (0)		FT- 100a[space][space]	No internal spaces allowed
Integer	Converts real numbers to integers		
Left	Displays from the left of the string, a specific number of characters according to user defined parameters	FT 1	In this example, 4 characters from the left of the string are displayed

Function	Description	Resulting String	Comments
Right	Displays from the right of the string, a specific number of characters according to user defined parameters	100 a[space][space]	In this example, 7 characters from the right of the string are displayed
Left Trim	Removes spaces to the left of the first character	FT - 100 a[space][space]	In this example any spaces to the left of the character F are removed from the string
Right Trim	Removes spaces to the right of the last character	FT - 100 a	In this example any spaces to the right of the character a are removed from the string

Standard Function Parameters

Some standard functions require additional parameters in their definitions:

Function	Parameters	Explanation
Substr	Start position	
	Length	The position of the first character of the sub- string from the start of the string to be displayed, and the number of characters to be displayed in the sub- string
RemSpace	Max. contiguous spaces	Maximum number of internal spaces allowed in the string
Left	Length	The number of characters to be displayed starting from the left of the string
Right	Length	The number of characters to be displayed starting from the right of the string

NOTES

- You cannot change the code of standard macro functions.
- When calculating the number of characters to be trimmed, remember that a space is counted as one character.

See also:

Working with User-Defined Macro Functions (on page 299)

Create a User-Defined Macro Function

Each user-defined macro function includes a set of selected standard functions. Use the following procedure to create user-defined macro functions at the database level.

1. Depending which module you have open, do one of the following:
 - In the Loop Drawings module, click **Tables > User Defined Macro Functions**.
 - In the Hook-Ups module click **Tools > User Defined Macros Functions**.
2. On the **User-Defined Macro Functions** dialog box, click **New** to open the **User- Defined Macro Function Properties** dialog box.
3. Type a name and macro function abbreviation for the new macro function, and, if needed, a description.

CAUTION If you want to use macro function abbreviations in loop block macros, make sure the macro function abbreviation is different from an instrument type abbreviation, which is a required segment in the loop macro function string. For details about loop blocks and loop block macro structure, see *Loop Blocks* (on page 256).
4. To superpose a standard function, do the following:
 - a. Click **Insert** to open the **Standard Macro Functions** dialog box.
 - b. Select a standard function and click **OK**.
 - c. If you selected a standard function that can accept parameters, specify appropriate values in the fields. For details, see *Standard Functions* (on page 300).

TIP The user-defined macro function operates in the macro string according to the order of the standard functions superposition. Use **Move Up** and **Move Down** to change the order of the standard functions.
5. Click **OK** to confirm your changes and close the **User-Defined Macro Function Properties** dialog box.
6. Do one of the following:
 - Select **Set macro function as default** to apply the specified macro function to all existing macros.
 - Clear **Set macro function as default** to be able to associate the specified macro function with a specific macro.

Modify a User-Defined Macro Function

1. Depending which module you have open, do one of the following:
 - In the Loop Drawings module, click **Tables > User Defined Macro Functions**.
 - In the Hook-Ups module click **Tools > User Defined Macros Functions**.
- 2.
3. On the **User-Defined Macro Functions** dialog box, select the row with the macro function you want to modify, and click **Properties** to open the **User-Defined Macro Function Properties** dialog box.
4. Modify the macro function properties as desired.
5. To superpose or change a standard function, do the following:
 - a. Click **Insert** to open the **Standard Macro Functions** dialog box.
 - b. Select a standard function and click **OK**.

- c. If you selected a standard function that can accept parameters, specify appropriate values in the fields. For details, see *Standard Functions* (on page 300).

TIP The user-defined macro function operates in the macro string according to the order of the standard functions superposition. Use **Move Up** and **Move Down** to change the order of the standard functions.

6. Click **OK** to confirm your changes and close the **User-Defined Macro Function Properties** dialog box.
7. Do one of the following:
 - Select **Set macro function as default** to apply the specified macro function to all existing macros.
 - Clear **Set macro function as default** to be able to associate the specified macro function with a specific macro.

Delete a User-Defined Macro Function

1. In the Loop Drawings or Hook-Ups module, click **Tables > User Defined Macro Functions**.
2. On the **User-Defined Macro Functions** dialog box, select the row with the macro function you want to delete.
3. Click **Delete**.

See Also

Modify a User-Defined Macro Function (on page 302)

Using External Macro Data Sources

You can use macros from an external source with the macros in your domain. This means that macro sources that are both external and internal in relation to SmartPlant Instrumentation are used when you generate a loop drawing.

Working with external macro sources includes the following steps:

1. Establishing a connection to an external data source from which SmartPlant Instrumentation can retrieve macro data.
2. In the **Domain Explorer**, assigning the connected external data source to the loop numbers for which you want to generate loop drawings.

IMPORTANT Before you start working with external macro data sources, make sure that the external source data naming and structure conventions comply with the SmartPlant Instrumentation macro conventions. For details, see *External Macro Structure Conventions* (on page 304) and *External Macro Naming Conventions* (on page 304).

NOTE Macros in a CAD application, that you want to solve with a general signal in SmartPlant Instrumentation, must have the general signal name (as defined in the SmartPlant Instrumentation **Local Signal** dialog box) added as a prefix to the CAD application macro. For example, for the CAD macro **PNL_NAME.1.4**, to be recognized by SmartPlant Instrumentation and solved with the general signal called **GENERSIG1**, you add the general signal name to the CAD macro as a prefix in the CAD application: **GENERSIG1.PNL_NAME.1.4**.

External Macro Naming Conventions

The macro names for external macros in a drawing must be identical to the names in the external database file or table.

NOTES

- To be able to use an external macro source, you need to establish a connection to the macro source in the **Define External Macro Source** dialog box.
- In your target drawing blocks or symbols, you need to add the **E_** prefix to the name of the macro that you want to use to retrieve data from the external macro source.

For example, if a drawing contains an external macro named **PROCESS** which is used to retrieve the string **Flow**, the macro label must be written as **E_PROCESS**. This means that after generation, every occurrence of the label **E_PROCESS** in the drawing will be replaced with the value **Flow**.

External Macro Structure Conventions

SmartPlant Instrumentation has set a few basic requirements regarding the structure of external data sources from which you add macros to your loop drawings. Therefore, before you use an external macro make sure that its structure complies with SmartPlant Instrumentation conventions. The structure has to include the tag number (cmpnt_name) as the first (leftmost) column, and occupying the rest of the columns, the desired macros and their retrieval values.

The following table is an example of an external source structure (the table headers below exemplify column names):

cmpnt_name	macro1	macro2	macro3
108-PI -2212	Pressure	Bar	Indicator
108-FE -2225	Flow	Ft3/s	Element

In the above example, using the macro **macro1** in a loop drawing generated for instrument tag 108-PI-2212, will display the retrieved value **Pressure** instead of the **macro1** string. You can create the above structure by using any text editing application (for Text format files), dBase (*.dbf) editing tools, or any other appropriate database editing utility. For further details, see the user's guide for your database platform.

IMPORTANT The name of the first (leftmost) column name must always be **cmpnt_name** (representing the tag number).

Connecting to an External Data Source

The first stage is connecting to the appropriate source data form whose macro data you want to use. After connecting to the database or the database file you can generate CAD drawings using the retrieved macros as required.

You can connect to any of the following database platforms and files:

- Oracle (only if you selected Oracle as your current database platform during Setup or you installed an Oracle client manually)
- Microsoft SQL Server (only if you selected SQL Server as your current database platform during Setup or you installed a SQL Server client manually)
- Sybase Adaptive Server Anywhere (only if you selected Sybase Adaptive Server Anywhere as your current database platform during Setup or you installed a Sybase Adaptive Server Anywhere client manually)
- Microsoft Access
- *.dbf
- ASCII delimited files
- Excel worksheets
- Text files

IMPORTANT The external database file with source macros must not reside in the SmartPlant instrumentation home folder.

NOTE You can connect to database platforms (Oracle, SQL Server or Sybase Adaptive Server Anywhere) other than your current database platform but you have to comply with the external database platform requirements. For example, if your current database platform is Sybase Adaptive Server Anywhere and you want to import data from Oracle, you must have an Oracle client installed and running on your computer. See Internal Setup Utility to learn how to configure your environment to connect to database platforms other than your current database platform.

See Also

Connecting to Database Platforms Other Than ODBC (on page 306)

Connecting to an ODBC Profile (on page 306)

Prerequisites for Connecting to a dBase ODBC Profile

You can generate loop drawings in SmartPlant Instrumentation using your dBase (.dbf) external macro source. Prior to connecting to dBase (.dbf) external macro source, you need to create a dBase ODBC profile in accordance with the driver requirements of your Windows system.

The following prerequisites apply:

- In your ODBC Data Source Administrator, select the Microsoft dBase Driver. For details, see Configure ODBC Data Source for DBF Files.
- Create an ODBC profile using your Windows ODBC Data source Administrator. For further details, see the user's guide for your database platform.

Connecting to an ODBC Profile

This procedure describes the steps required to connect to a database file (for example, .dbf, ASCII, and so forth) using the ODBC profile. This way you can generate your CAD drawings using macros from an external macro source.

The following file types are available:

- dBase (.dbf)
- Excel worksheets
- ASCII delimited files
- Text files

NOTE If you want to connect to dBase (.dbf file), you must create an ODBC profile in accordance with your Windows system. For more details, see *Prerequisites for Connecting to a dBase ODBC Profile* (on page 305) and *Configure ODBC Data Source for DBF Files*.

Connect to an ODBC Profile

1. Click **Actions > Define External Macro Source**.
2. On The **Define External Macro Source** dialog box, from the **Database type** list, select **ODBC**.
3. From the **Data source** list, select the source database file profile, for example, .dbf, ASCII files, and so forth.
4. Click **Browse** and select macro source file.

See Also

Connecting to an External Data Source (on page 305)

Connecting to Database Platforms Other Than ODBC

This procedure describes the steps required to connect to a **database platform** such as Oracle or SQL Server. To connect to an external database platform, you must have access rights to the database server as well as the appropriate database drivers. You must also have the appropriate settings in the configuration files (for example, .ini files) and in the Windows registry.

Connect to Database Platforms Other Than ODBC

1. Click **Actions > Define External Macro Source**.
2. In The **Define External Macro Source** dialog box, from the **Database type** list, select the source database platform: Oracle or SQL Server.
3. In the **Server** box, type the name of the server that you use to connect to the database server:
 - <alias name> — if you connect to Oracle
 - <server name> — if you connect to SQL Server
4. In the **Database schema logon name** box, type the logon name required to connect to the source domain in the database server.
5. In the **Database schema logon password** box, type the logon password required to connect to the source domain in the database server.

Assigning an External Macro Data Source

This procedure enables you to assign an external macro data source to one or more loop numbers.

NOTES

- To be able to use an external macro source, you need to establish a connection to the macro source in the **Define External Macro Source** dialog box.
 - In your target drawing blocks or symbols, you need to add the **E_** prefix to the macros which you want to use to retrieve data from an external macro source.
1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.
 2. Do one of the following:
 - In the tree view pane, right-click a desired loop.
 - **Items** pane, select and right-click several loops.
 3. On the shortcut menu, click **Actions > Assign External Macro Source**.
 4. Under the **Macro Source** column, do one of the following:
 - If your macro source is an ODBC profile, type the exact path and the file name of the source database file.
 - If your macro source is a specific table on Oracle or SQL Server, you need to type the exact table name.

TIP If you want to assign the same macro source to all the loop numbers displayed in the **Assign External Macro Source** dialog box, select a row with the desired macro source and then select **Apply same source to all displayed loop numbers**.

See Also

Using External Macro Data Sources (on page 303)

Connecting to an External Data Source (on page 305)

MicroStation and SmartPlant Instrumentation

MicroStation is a three-dimensional CAD software package which includes features to allow the interchange of design data with SmartPlant Instrumentation. SmartPlant Instrumentation supplies the design data, and generates a schematic drawing associated with a specific loop, component or block by means of MicroStation.

NOTES

- SmartPlant Instrumentation use of the term **block** corresponds to the term **cell** in MicroStation.
- This section explains how MicroStation works with SmartPlant Instrumentation. All the topics in this section require a basic level of familiarity with MicroStation. Refer to the MicroStation user manual for detailed explanations of MicroStation functionality.

System Requirements

The following minimum hardware and software requirements must be met before installing MicroStation on your computer:

- 16 MB RAM in addition to SmartPlant Instrumentation installation requirements.
- 75 MB of free hard disk space.

Refer to the Getting Started section of your MicroStation user manual for more detailed requirements of the MicroStation software package.

Set MicroStation for SmartPlant Instrumentation

Once MicroStation has been successfully installed, a number of modifications still have to be made to ensure compatibility with SmartPlant Instrumentation.

Define MicroStation Settings in SmartPlant Instrumentation

To generate CAD loop drawings using MicroStation, the temporary folder path setting on the **General** page of the SmartPlant Instrumentation **Preferences** dialog box must not exceed fifty two characters. This means that you cannot use the default path C:\Program Files\SmartPlant\Instrumentation\TEMP. The same restriction applies to the CADFunc folder path you define on the **CAD File Locations** page for loop drawings or hook-up drawings on the **Preferences** dialog box. A longer path prevents MicroStation from generating or displaying drawings or cells from SmartPlant Instrumentation. If your CAD function folder path is longer than allowed, you need to copy the CADFunc folder and its content to another location, for example, C:\CAD\CADFunc.

Define MicroStation Settings in MicroStation

The following modifications need to be done using the options on the menu bar of the main **MicroStation** window.

1. Click **Workspace > Preferences**.
2. On the **Preferences** dialog box, under **Category**, select **Memory Usage** and ascertain that your settings are as follows:

Setting	MicroStation SE	MicroStation J
Max. Element Cache	8000	10240
Resource Cache	24	1024
Undo Buffer	256	2048
Font Cache	30	256
Conserve Memory	Cleared	Cleared
Disable OLE Automation	Cleared	Cleared

IMPORTANT When working with MicroStation, clear **Conserve Memory** check box.

3. Under **Category**, select **Operation** and ascertain that your settings are as follows:

Setting	Value
Locate Tolerance	10
Pointer Size	Normal
Pointer Type	Orthogonal
Display Levels	Names
Immediately Save Design Changes	Selected
Save Settings on Exit	Cleared
Compress Design on Exit	Cleared
Enter into Untitled Design	Selected
Reset Aborts Fence Operations	Selected
Level Lock Applies for Fence Operations	Selected
Use Semaphore File for Locking	Cleared

TIP When working with MicroStation, the **Immediately Save Design Changes** check box must be selected to enable viewing or generation of loop drawings if MicroStation was not previously launched.

4. Under **Category**, select **Tags** and ascertain that your settings are as follows:

Setting	Value
Prompt on Duplicate Tag Sets	Cleared
Use Design File Tag Sets by Default	Cleared
Place Tags in Same Graphic Group	Cleared

5. When finished, click **OK** to return to the main **MicroStation** window.

SECTION 36

Working with MicroStation

This section contains topics that relate to procedures within Microstation.

In This Section

Create Tag Sets (MicroStation)	311
Export Tag Sets (MicroStation)	312
Import a Tag Set (MicroStation).....	312
Link a Tag to a Drawing Element (MicroStation)	312
Create a Cell Library (MicroStation)	313
Fence Elements for a Cell (MicroStation).....	313
Define the Cell Origin (MicroStation)	314
Define the Coordinates of the Cell Origin (MicroStation)	314
Add a Cell to the Cell Library (MicroStation)	314
Preview a Cell (MicroStation)	314
Display a Cell on the Desktop (MicroStation).....	315
Generating a Report Using MicroStation.....	315

Create Tag Sets (MicroStation)

In MicroStation you group tags into tag sets. Use this procedure when you need to create a new tag set from scratch. To save time it is recommended that you duplicate and rename tag sets instead of creating new ones. You can save tag sets to a tag set library file and then copy them to other files as desired.

1. In the main MicroStation window, click **Element > Tags > Define**.
2. In the **Tag Sets** window, under **Sets**, click **Add**.
3. On the **Tag Set Name** dialog box, type the desired name, for example **instrument**.
4. Click **OK** to accept the value and reopen the **Tag Sets** window.
5. Repeat the above steps to add another tag set named **loop**.
6. In the **Tag Sets** window, under **Sets**, select **instrument**.
7. Under **Tags**, click **Add**.
8. On the **Define Tag** dialog box, type **tag_loc**. This value represents the tag location.
9. Repeat the above steps to add another property named **tag_num**.
TIP The text in the **Default Tag Value** pane must be typed in upper case.
10. In the **Tag Sets** window, under **Sets**, select **loop**.
11. Under **Tags**, add the following properties:
 - **loop_func** (Loop Function)
 - **loop_type** (Loop Type)

Export Tag Sets (MicroStation)

Exporting Tag Sets allows you to save new tag sets in a library file. This makes it more convenient to import the exported tag set.

1. In the **Tag Sets** window, select the **Set** (for example, **instrument**) to export.
2. Click **File > Export > Create**.
3. On the **Export Tag Library** dialog box, click **OK** to return to the **Tag Sets** window.
4. Repeat the previous steps to create a tag library for the loop set.

NOTE The next time you open a MicroStation session, select **Import** to display the latest tag set definition.

Import a Tag Set (MicroStation)

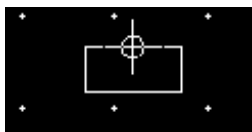
This procedure explains how to retrieve a previously created tag set from a tag library file.

1. In the **Tag Sets** window, click **File > Import**.
2. On the **Open Tag Library** dialog box, select the desired library and click **OK** to open the **Import Sets** dialog box.
3. Click **OK** to reopen the **Tag Sets** window with the set and its associated tags displayed.
4. To add another set, to the **Tag Sets** window, repeat steps 1 and 2, but this time select **instrmnt.tlb** on the **Open Tag Library** dialog box.

Link a Tag to a Drawing Element (MicroStation)

This procedure explains how to create a drawing element and associate it with the tag definitions that you defined in the **Define Tag** window.

1. In the main **MicroStation** window, click **Tools > Main > Tags**.
2. In the **Tags** window, select, for example, the **Place Block** tool from the **Main tool palette** to draw a rectangle on the MicroStation desktop.
3. Select **Attach tags to an element** to open the **Attach Tags** dialog box.
4. On the **Attach Tags** dialog box, place the cursor on the rectangle that you drew in step two:



5. Double click the cursor to open the **Attach Tags** dialog box.

TIP If you want to display another tag set, such as the loop tag set, open the **Attach Tags** dialog box and repeat steps 3 - 5.

6. Click **OK** to display the tags on the desktop.

7. Move the tags, without clicking, to the desired location. Below is a typical example.



8. Click the left mouse button when the tag position is appropriate.
The tag is now associated with the design element (the rectangle in this case).

Create a Cell Library (MicroStation)

This procedure explains how to create a cell library to which you can add cells.

1. In the main **MicroStation** window, click **Element > Cells** to open an unnamed **Cell Library** window.
2. Click **File > New** to open the **Create Cell Library** dialog box.
3. Click **OK** to reopen the **Cell Library** window.
4. Drag the **Cell Library** window as far off the desktop as possible and store it for future use. The window will be recalled later when you add a cell to it.

Fence Elements for a Cell (MicroStation)

You can fence elements to globally perform actions on them in batch mode. This allows you to identify the cell in a cell library to be recalled as many times as needed. This enables you for example to edit the cell, or combine it with other cells to create a larger design.

1. Set all the elements on the desktop, and select the fence icon on the main tool palette.



2. Fence the desired elements.



NOTE Fencing an area automatically opens the **Cell Library** window. Prior to saving a cell in the cell library you need to define the cell origin.

Define the Cell Origin (MicroStation)

You define a cell origin to set the position of a graphic object in a design. This will subsequently and automatically entail the **Create** action in the **Cell Library** window.

1. Select the **Define Cell Origin** icon from the Main tool palette.
2. Click the cursor on the lower left corner of the desktop.

NOTE Now that the cell origin is defined, you have to accurately set the cell origin coordinates (for details, see *Define the Coordinates of the Cell Origin* (see "Define the Coordinates of the Cell Origin (MicroStation)" on page 314)).

Define the Coordinates of the Cell Origin (MicroStation)

This procedure explains how to generate the cell origin coordinates for a graphic element placed on the desktop.

1. With the cell origin defined (typically, on the lower left corner of the desktop), click  to open the **AccuDraw** window.
2. Set the X and Y coordinates to 0.0000.
3. Select all the check boxes.

Add a Cell to the Cell Library (MicroStation)

1. Open the **Cell Library** window.
2. Click **Create** to open the **Create New Cell** dialog box.

NOTE Both **Name** and **Description** must be defined otherwise the software does not perform the create action.
3. Click **Create** to display the new cell name and description in the data window of the **Cell Library** window.

Preview a Cell (MicroStation)

When several cells are listed in the cell library, it is recommended that you preview a cell to make sure that you have selected the correct one. This procedure explains how to generate a thumbnail view of the cell.

1. Open the appropriate cell library.
2. Click the cell that you want to preview.

*A thumbnail view of the cell appears in the **Print Preview** pane.*

Display a Cell on the Desktop (MicroStation)

This procedure explains how to take a previewed cell and place it on the desktop. This may also be useful when you want to relocate a cell to achieve a different design than the current one.

1. Open the **Cell Library** window, click a cell to preview it, and then click **Placement**.
2. On the Main tool palette, click the **Place Active Cell** icon.

TIP A tool from the **View Control** bar may be needed to display your Cell.

3. Move the mouse to the desired location on the desktop.
4. Click to pin the cell to a selected location.
5. If you need another copy of the same cell, repeat steps 3 and 4.
6. When done, right click.

TIP You can also add cells from other libraries, print/plot your design, and so forth. Refer to the MicroStation user manual to learn more about the available MicroStation options.

Generating a Report Using MicroStation

You can generate SmartPlant Instrumentation reports using MicroStation from either SmartPlant Instrumentation or MicroStation. When you generate the report from the MicroStation environment you need to manually open the desired report file using the **Open** dialog box in MicroStation. However, the recommended method is to generate the reports from the SmartPlant Instrumentation Loop Drawings module (for details, see *CAD Loop Drawing Generation: An Overview* (see "CAD Loop Drawing Generation" on page 276)).

Hook-Ups in SmartPlant Instrumentation

In SmartPlant Instrumentation, you can create, view and edit installation details for all prepared instrument drawings (hook-ups) within a domain. A hook-up is a graphical representation of an assembly drawing — a drawing prototype.

You can perform various actions in the **Reference Explorer**, the **Domain Explorer**, and the Hook-Ups module.

In the **Hook-Up Item Libraries** folder of the **Reference Explorer**, you define your item libraries, create and manage hook-up items. You need to set a specific library as the active item library, which becomes your <plant> default library. The completeness of this library is up to the user.

In the **Hook-Ups** folder of the **Domain Explorer**, which depicts the hierarchy of hook-up data, you can organize hook-ups in hook-up types and associate hook-ups with instrument tags and hook-up items. You can determine the name of each hook-up and hook-up type based on your specific requirements. After that, you can generate hook-up drawings using the Enhanced Report Utility or your CAD application.

Using the Hook-Ups module options, you generate various hook-up reports and a Bill of Material, define hook-up item manufacturers, make hook-up macro definitions, and so forth.

Hook-Ups Module Window

This window allows you to use general hook-up options, which do not depend on specific item selection in the Domain Explorer. In this window, the following menu commands are available:

Menu Command	Description
Associations > Default General Blocks	Allows you to select default border and logo blocks to appear in all hook-up drawings that you generate. For details, see Default General Blocks Dialog Box.
Associations > Update Paths for Existing Drawings and Blocks	Allows you to update paths globally for block files associated with tag numbers and for generated hook-up drawings. The paths that you specify do not overwrite the default paths specified for new block files and output drawings on File Locations page of the Preferences dialog box. For details, see Update Paths for Existing Drawings and Blocks Dialog Box.
Reports	Allows you to generate all reports that are available in the Hook-Ups module, for example, a Bill of Material , the Library Items report, and so forth. For details, see <i>Hook-Up Reports</i> (on page 345).
Tables > Macro Definitions	Allows you to create or modify user-defined macros for use with hook-up drawings only. For details, see Macro Definitions Dialog Box.
Tables > Item Manufacturers	Allows you to manage hook-up item manufacturer definitions and their descriptions. For details, see Item Manufacturers Dialog Box.
Tools > User-Defined Macro Functions	Allows you to define or modify macro functions that can then be used with user-defined macros for hook-up or loop drawings. For details, see User-Defined Macro Functions Dialog Box.

Hook-Up Items and Libraries

A hook-up drawing is made up of individually marked parts called items. These items have to be purchased. The service that a Bill of Material provides you with depends upon how completely you fill in the hook-up item library. The minimum amount of data required to provide is the item number. Entering more data is left to your discretion.

SmartPlant Instrumentation allows you to define a large number of hook-up items and group them in item libraries and sub-libraries. You create and manage item libraries in the **Reference Explorer**. A new item library that you create automatically contains a default sub-library, which you can use for assigning hook-up items if you do not work with pipe specs. For more information on implementation of pipe specs in SmartPlant Instrumentation, see Pipe Specs. When defining a new item in SmartPlant Instrumentation, you can enter a number of hook-up item properties, such as item size, model, material, storage number, unit of measure, and so forth. Also, it is possible to define hook-up item manufacturers in the Item Manufacturers the supporting table and then assign the appropriate manufacture to the item.

You must set one item library as the active item library for your <plant>. After you create hook-up items in this library, you can use the **Domain Explorer** options to associate the appropriate sub-library with hook-ups, and then, associate specific items with these hook-ups. After making the associations, you can generate Hook-Up Item List reports and a Bill of Material. In the Bill Of Material, you can sort the items by the item order. After associating hook-up items with a hook-up, the software generates new sequence numbers automatically and allows you to sort the items by their order of their assignment to a hook-up.

In the **Reference Explorer**, hook-up items are displayed in the **Hook-Up Item Libraries** folder, under sub-libraries. In the **Domain Explorer** hook-up items are displayed in the **Hook-Ups** folder, under hook-ups. Hook-up items are indicated with the  icon.

Hook-Up Item Libraries and Sub-Libraries

An item sub-library is a term which refers to a part of an item library and contains a specific set of hook-up item data and associations. You create and manage item libraries in the **Reference**

Explorer, where item libraries are indicated with the  icon. An item library can contain any number of user-defined sub-libraries that you can use when working with pipe specs. A new item library that you create automatically contains a default sub-library, which you can use for assigning hook-up items if you do not work with pipe specs. For more information on implementation of pipe specs in SmartPlant Instrumentation, see Pipe Specs. Sub-libraries are indicated with the  icon.

You must set one item library as the active item library for your <plant>. After you create hook-up items in this library, you can associate the appropriate sub-library with hook-ups, and then, associate specific items with these hook-ups. In the **Reference Explorer**, the active item library is indicated with the  icon.

Workflow for Creating Hook-Up Items in the Reference Explorer

This workflow describes the steps needed to create a hook-up item in the reference explorer.

1. In the Reference Explorer, **Hook-Up Item Libraries** folder, under **Default Sub-Library**, create a hook-up item. For more information, see *Create a Hook-Up Item* (on page 321).
2. In the Reference Explorer, **Hook-Ups Library**, create a user-defined sub-library. For more information, see *Create User-Defined Sub-Libraries* (on page 320).
3. If pipe specs are to be associated with the hook-up item, then you must first define or modify the required pipe spec. For more information, see Define or Modify Pipe Specifications. see the *SmartPlant Instrumentation Basic Engineering User's Guide > Instrument Index Module > Managing Lines > Define or Modify Pipe Specifications*.
4. After defining the pipe specs, you must associate them with the user-defined sub-library created previously. For more information, see Associate Pipe Specs with Sub- Libraries. see the *SmartPlant Instrumentation Basic Engineering User's Guide > Specifications > Pipe Specs > Associate Pipe Specs with Sub-Libraries*.
5. Assign the hook-up item to the user-defined sub-library. For more information, see *Associate Items with User-Defined Sub-Libraries* (on page 322).
6. Add any macros that are required. For more information, see *Create a New Hook-Up Macro Definition* (on page 298).
7. Generate your hook-up drawing. For more information, see the following topics:
 - *Generate CAD Hook-Up Drawings* (on page 339)

- *Generate Enhanced Hook-Up Drawings* (on page 340)
- *Generate Hook-Up Drawings in Mixed Mode* (on page 341)

See also

Hook-Ups in SmartPlant Instrumentation (on page 315)

Hook-Up Items (see "*Hook-Up Items and Libraries*" on page 316)

Hook-Up Drawing Generation (on page 331)

Managing Hook-Up Items and Libraries Common Tasks

These tasks deal with creating and managing hook-up item libraries, sub-libraries, and hook-up items in the **Reference Explorer**.

Create an Item Library

This procedure describes how to create a new item library. An item library contains fittings and associations created on a domain level. Therefore, when you enter a domain, you can have any number of relevant libraries. The number of libraries, names, and completeness are all up to your requirements. An item library can also contain any number of user-defined sub-libraries that you can use when working with pipe specs. A new item library that you create automatically contains a default sub-library, which you can use for assigning hook-up items if you do not work with pipe specs. For more information, see *Create an Item Library* (on page 319).

Set an Item Library As the Active Item Library

Use this procedure to set a specific item library as the active item library for the current <plant>. You must set a specific library as the active item library to be able to create hook-ups in the current <plant> and assign the library items to hook-ups. The information is filled in to pick out parts of a specific hook-up. You can also enter description, material, rating, and so forth, to provide as much information as possible for the Bill of Material that the software creates based on this data. For more information, see *Set an Item Library As the Active Item Library* (on page 320).

Assign the Same Active Item Library to More Than One Plant

Use this procedure if you need to assign an existing active item library to more than one <plant> in your domain. For more information, see *Assign the Same Active Item Library to More Than One Plant* (on page 320).

Create User-Defined Sub-Libraries

Use this procedure to create user-defined sub-libraries in the library set as the active item library. You can also assign pipe specs to user-defined sub-libraries. For more information, see *Create User-Defined Sub-Libraries* (on page 320).

Create a Hook-Up Item

Use this procedure to create a hook-up item under the default or user-defined sub-library. For more information, see *Create a Hook-Up Item* (on page 321).

Create Hook-Up Items in Batch Mode

Use this procedure to create several hook-up items in the default sub-library of a specific hook-up library. For more information, see *Create Hook-Up Items in Batch Mode* (on page 321).

Sort Hook-Up Items in a Sub-Library

Use this procedure to sort the hook-up items defined in a specific sub-library. For more information, see *Sort Hook-Up Items in a Sub-Library* (on page 322).

Associate Items with User-Defined Sub- Libraries

Use this procedure to assign items that exist in the default sub-library to a user-defined sub-library. For more information, see *Associate Items with User-Defined Sub- Libraries* (see "Associate Items with User-Defined Sub-Libraries" on page 322).

Remove Items from a User-Defined Sub- Library

Use this procedure to remove one or more items from a user- defined sub-library. When removing the items, the software only dissociates them from that sub-library but does not delete them from the parent hook-up item library. For more information, see *Remove Items from a User-Defined Sub- Library* (see "Remove Items from a User-Defined Sub-Library" on page 323).

Delete Hook-Up Items

Use this procedure to delete one or more hook-up items from a specific hook-up item library. For more information, see *Delete Hook-Up Items* (on page 323).

Delete Hook-Up Sub-Libraries

Use this procedure to delete an empty user-defined sub- library from the parent hook-up item library. For more information, see *Delete Hook-Up Sub-Libraries* (on page 324).

Create an Item Library

1. Press F8 to open the **Reference Explorer**, and right-click the **Hook-Up Item Libraries** folder.
2. On the shortcut menu, click **New > Item Library**.
3. Type a unique name for the item library and an optional description.
4. Click **OK**.

TIPS

- On creating the item library, the software automatically creates the default sub- library under the library.
- You can create a hook-up library by duplicating an existing library. When duplicating the library, the software also duplicates all the sub-libraries and the items that are associated with the sub-libraries. If a library contains a large number of sub-libraries and hook- up items, the duplication process might take time.

Set an Item Library As the Active Item Library

1. In the **Reference Explorer**, expand the **Hook-Up Item Libraries** folder.
2. Right-click a library that you require.
3. On the shortcut menu, click **Actions > Set As Active item Library**.

TIPS

- You can only set one library as the active item library for a <plant>. If another item library has been set as the active item library for the current <plant>, the software dissociates the hook-ups from the previous active library and assigns the default sub-library of the new active library to all the hook-ups in your <plant>.
- In the **Reference Explorer**, the library icon changes from  to .

Assign the Same Active Item Library to More Than One Plant

1. Open a <unit> in the target plant.
2. In the **Reference Explorer**, expand the **Hook-Up Item Libraries** folder.
3. Right-click the library that has been set as the active item library in another <plant>.
TIP Since you can only set one library as the active item library for a specific <plant>, the **Hook-Up Item Libraries** folder does not indicate any active item libraries specified for other <plants>.
4. On the shortcut menu, click **Actions > Set As Active item Library** and change the library icon from  to .
5. Repeat this procedure for every plant to which you want to assign the same active item library.

Create User-Defined Sub-Libraries

1. Press F8 to open the **Reference Explorer**, and expand the hierarchy to the item library level in the **Hook-Up Item Libraries** folder.
2. Right-click the item library.
3. On the shortcut menu, click **New > Item Sub-Library**.
4. Type a unique name for the item sub-library and an optional description.

TIP You can click **Pipe Specs** and assign pipe specs to the sub-library you are creating. If you have not defined pipe specs yet, you can create the sub-library first and assign pipe specs at a later stage.

5. Click **OK** to create the sub- library.

TIP It is possible to create more than one sub-library by duplicating several existing user-defined sub-libraries that you can select in the **Items** pane. To display the sub-libraries in the **Items** pane, in the tree view pane, select the parent item library. When duplicating a user-defined sub- library, the software also duplicates all of the hook- up items that exist in that sub-library. You cannot duplicate the default sub-library.

Create a Hook-Up Item

1. Press F8 to open the **Reference Explorer**, and expand the hierarchy to the item library level in the **Hook-Up Item Libraries** folder.
2. Double-click the item library to display the item sub-libraries.
TIP The default sub-library exists in every library that you create. In addition, you can create user-defined sub-libraries if you want to associate pipe specs to hook-ups.
3. Right click the default or user-defined sub-library.
4. On the shortcut menu, click **New > Hook-Up Item**.
5. Type a unique name for the hook- up item and define other properties as needed.
6. Click **OK** to create the hook-up item.

TIP If you created a new item in a user-defined sub-library, this item automatically appears in the default sub-library too.

Create Hook-Up Items in Batch Mode

1. Press F8 to open the **Reference Explorer**, and expand the hierarchy to the item library level in the **Hook-Up Item Libraries** folder.
2. Double-click the item library to display the item sub-libraries.
TIP The default sub-library exists in every library that you create. In addition, you can create user-defined sub-libraries if you want to associate pipe specs to hook-ups.
3. Right-click the default sub-library.
4. On the shortcut menu, click **Actions > Batch Item Creation**.
5. Under **Number of items to create**, use the spinner to enter the number of the hook- up items you want to create under the default sub- library.
6. In the **Prefix** box, type an optional prefix you want to assign to the batch of the items you are about to create.
7. In the **Start from number** box, type the starting number you want to append to the name of the first item in the batch.
TIP If you leave the **Start from number** box empty, after you click **OK**, the software automatically appends **0** (zero) to the name of the first item in the batch.
8. In the **Increment by** box, use the spinner to change the incrementing value as needed.
TIP The default value in the **Incremented by** box is **1**. When using the default value, to ensure item name uniqueness within the current default sub-library, the software increments every subsequent item number in the batch of items by one digit. you can change this value as needed. For example, if your want to create three items with prefix **MyItem**, the **Start from number** value **5**, and the **Increment by** value **2**, the software creates the following items: **MyItem5**, **MyItem7**, and **MyItem9**.
9. Click **OK** to open the **Hook-Up Item Properties** dialog box, where you can define properties for each item and complete the item creation.

Sort Hook-Up Items in a Sub-Library

1. In the tree view of the **Reference Explorer**, select and right-click a sub-library.
2. On the shortcut menu, click **Sort**.
3. From the **Item type** list, select **Hook-Up Item**.
4. Under **Available Property**, select a property you want to use for sorting hook-up items, for example **Item Number**.
5. Drag this property to the **Sort Property** pane, select the check box and click **OK**.

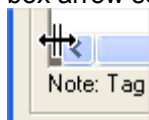
TIP The software sorts the items that start with a numeric character first. For example, if you sort by item number items whose numbers are **5, 1, A, 4, AA, BB, A, AB, 2, 3**, these items appear in descending order as follows: **1, 2, 3, 4, 5, A, AA, AB, B**.

Associate Items with User-Defined Sub-Libraries

1. In the **Hook-Up Item Libraries** folder of the **Reference Explorer**, do one of the following:
 - In the tree view pane, expand the hierarchy of an item library and right-click a user-defined sub-library.
 - In the tree view pane, select an item library and then, in the **Items** pane, select one or more user-defined sub-libraries.
2. On the shortcut menu, click **Actions > Associate Items with Sub-Libraries**.

TIPS

- On the dialog box that opens, the software displays all the items available in the default sub-library of the library you selected. You can set a filter to display only those items that you require for association. In the filter that you require in not available in the **Filter available items** list, you can click  and define a new filter.
- In the dialog box panes, you can double-click a column header in the to sort the items in ascending order. Double-click again to sort the items in descending order.
- In the dialog box panes, there is a vertical bar that you can drag to the right and view your data in a split display. To find the vertical bar, place the cursor to the left of the left scroll box arrow so that the cursor changes its shape as shown:



3. Under **Available items**, select one or more hook-up items and do one of the following:
 - Click **Associate**.
 - Drag the selected items to the **Associated items** pane.
4. If you selected more than one user-defined sub-library in the **Reference Explorer**, click **Next** to display another sub-library and associate hook-up items as you require.

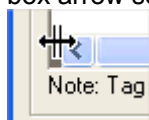
TIP You can associate the same items with more than one user-defined sub-library.

Remove Items from a User-Defined Sub-Library

1. In the **Hook-Up Item Libraries** folder of the **Reference Explorer**, do one of the following:
 - In the tree view pane, expand the hierarchy of an item library and right-click a user-defined sub-library.
 - In the tree view pane, select an item library and then, in the **Items** pane, select one or more user-defined sub-libraries.
2. On the shortcut menu, click **Actions > Associate Items with Sub-Libraries**.

TIPS

- On the dialog box that opens, the software displays all the items available in the default sub-library of the library you selected. You can set a filter to display only those items that you require for association. In the filter that you require in not available in the **Filter available items** list, you can click  and define a new filter.
- In the dialog box panes, you can double-click a column header in the to sort the items in ascending order. Double-click again to sort the items in descending order.
- In the dialog box panes, there is a vertical bar that you can drag to the right and view your data in a split display. To find the vertical bar, place the cursor to the left of the left scroll box arrow so that the cursor changes its shape as shown:



3. Under **Associated items**, select one or more hook-up items and do one of the following:
 - Click **Dissociate**.
 - Drag the selected items to the **Available items** pane.
4. If you selected more than one user-defined sub-library in the **Reference Explorer**, click **Next** to display another sub-library and dissociate hook-up items as you require.

Delete Hook-Up Items

1. Press F8 to open the **Reference Explorer**, and expand the hierarchy to the item sub-library level in the **Hook-Up Item Libraries** folder.
2. In the tree view pane, do one of the following:
 - Double-click the default sub-library from which you want to delete a specific item, and then select and right-click a specific item you want to delete.
 - Select the default sub-library from which you want to delete one or more items, and then, in the **Items** pane, select and right-click items you want to delete.
3. On the shortcut menu, click **Delete**.

TIPS

- The software only allows you to delete those items that are not associated with hook-ups.
- If you created user-defined sub-libraries, the software automatically deletes the items from all the sub-libraries that exist in the library. You can also perform item deletion from a user-defined sub-library.

Delete Hook-Up Sub-Libraries

1. Press F8 to open the **Reference Explorer**, and double-click the **Hook-Up Item Libraries** folder.
2. Select an item library to display its sub-libraries in the **Items** pane.
3. In the **Items** pane, select and right-click one or more user-defined sub-libraries.
4. On the shortcut menu, click **Delete**.

TIP You can only delete empty user-defined sub-libraries that exist in the library.

Managing Hook-Ups Common Tasks

These tasks deal with creating and managing hook-ups and hook-up types in the **Domain Explorer**. Also, these tasks describe various association procedures you need to perform, for example, association of instrument tags to hook-ups, association of items to hook-ups, and so forth.

Create a Hook-Up Type

Use this procedure to create a hook-up type at the domain level. A hook-up type is a common name of group of hook-ups that share distinguishing characteristics common to one kind of instrument. These characteristics set one hook-up type apart from another type, for example **Flow**, **Temperature**, **Level**, and so forth. You can only create one hook-up type at a time. For more information, see *Create a Hook-Up Type* (on page 326).

Create Hook-Ups

Use this procedure to add one or more hook-ups to existing hook-up types. Hook-ups that you create under hook-up types only appear in the current <plant>. A hook-up is a graphical representation of an assembly drawing — a drawing prototype. Adding a hook-up to a hook-up type is a prerequisite for assigning instruments tags to hook-ups. For more information, see *Create Hook-Ups* (on page 326).

Associate Hook-Up Types with Instrument Types

Use this procedure to assign one or more hook-up types an instrument type. Your database can contain a seemingly endless number of tag numbers. To make the task of attaching multiple instruments to a specific hook-up type manageable, use a filter that allows you to assign any instrument type in the current domain to your hook-up type.. For more information, see *Associate Hook-Up Types with Instrument Types* (on page 327).

Associate Instrument Tag Numbers with Hook-Ups

Use this procedure to assign one or more tag numbers to one or more hook-ups. The hook-ups can belong to the same or different hook-up types. For more information, see *Associate Instrument Tag Numbers with Hook-Ups* (on page 327).

Move Hook-Ups to Another Hook-Up Type

This procedure enables you to move all hook-ups from one or more hook-up types to another existing hook-up type. For more information, see *Move Hook-Ups to Another Hook-Up Type* (on page 328).

Change a Hook-Ups Sub-Library Association

Use this procedure to associate a hook-up with a user- defined sub-library or to change the existing hook-up sub- library assignment.

You can associate a hook-up with a sub-library when performing the following activities:

- Adding a new hook-up to a hook-up type — You can assign any new hook-up to any sub-library.
- Editing properties of an existing hook- up
- Duplicating a hook-up

When editing or duplicating hook-up properties, you can change the existing association of a hook- up sub-library under the following conditions:

- If the hook-up has associated instrument tags with pipe spec data, the target user- defined sub-library must also have pipe specs suitable for association of these tags.
- The target user-defined sub- library must contain all hook-up items that are associated with the hook-ups.

For more information, see *Change a Hook-Ups Sub-Library Association* (on page 328).

Associate Items with Hook-Ups

This procedure enables you to associate one or more items with hook-ups. You can only associate those items that exist in the sub-library assigned to the hook-ups that you select in the **Domain Explorer**. In hook-up drawings that you generate, the associated items are displayed as the identifying. For more information, see *Associate Items with Hook-Ups* (on page 329).

Dissociate Items from Hook-Ups

Use this procedure to dissociate one or more items from a hook-up. You must dissociate an item from a hook-up prior to deleting this item from the active item library. For more information, see *Dissociate Items from Hook-Ups* (on page 330).

Dissociate Tag Numbers from Hook-Ups

Use this procedure to dissociate one or more instrument tags from a hook-up. You must dissociate all instruments from a hook-up prior to deleting this hook-up. For more information, see *Dissociate Tag Numbers from Hook-Ups* (on page 330).

Delete Hook-Ups

This option enables you to delete a hook-up which is not associated with any tag numbers. If an association exists, first dissociate the tag numbers from this hook-up and then delete the hook-up. For more information, see *Delete Hook-Ups* (on page 331).

Delete Hook-Up Types

This option enables you to delete one or more hook-up types which are not associated with any hook-ups. To delete a hook-up type that contains hook-ups, first delete the hook-ups or move them to another hook-up type. For more information, see *Delete Hook-Up Types* (on page 331).

Create a Hook-Up Type

1. Press F7 to open the **Domain Explorer**, expand the hierarchy of any <plant> and right-click the **Hook-Ups** folder.
TIP Since hook-up types are defined at the domain level, any hook-up type that you create appears in the **Hook-Ups** folder of every <plant> that exists in the current domain.
2. On the shortcut menu, click **New > Hook-Up Type**.
3. In the **Hook-Up Type Properties** dialog box, type a unique hook-up type name and an optional description.
4. Click **OK**.

Create Hook-Ups

Prerequisite

- Set a specific hook-up item library as the active item library for the current <plant>. For details, see *Set an Item Library As the Active Item Library* (on page 320).
1. Press F7 to open the **Domain Explorer**.
 2. Expand the item hierarchy of the current <plant> and do one of the following:
 - Right-click the **Hook-Ups** folder.
 - Double-click the **Hook-Ups** folder and then, in the **Items** pane, select and right-click the hook-up type to which you want to add a new hook-up.
 3. On the shortcut menu, click **New > Hook-Up**.
 4. If you right-clicked the **Hook-Ups** folder, from the **Hook-up type** list, select the target hook-up type.
 5. In the appropriate boxes, enter a name for the new hook-up and a description.
NOTE If the same name is used for more than one hook-up then the description must be unique.
 6. Type a unique name for the item sub-library and an optional description.
 7. From the **Item sub-library** list, select a sub-library to which you want to assign the hook-up.
TIP If you do not work with user-defined sub-libraries, the software assigns the hook-up to the default sub-library automatically. If you work with user-defined sub-libraries, we recommend that you read the topic *Change a Hook-Ups Sub- Library Association* (see "Change a Hook-Ups Sub-Library Association" on page 328).
 8. Beside **Drawing block file name and path**, click **Browse** to navigate to the drawing block file name and path.
TIPS
 - Click **View** to open the selected drawing in the CAD application installed on your computer.
 - The path setting that appears in the **Generation output path** box is the path that you specified in the **Generate Hook-Up Drawings** dialog box. The software only displays the path if you already generated a hook-up drawing for the source hook-up.
 9. Click **OK** to create the hook-up.

TIP It is possible to create more than one hook-up by duplicating several existing hook-ups that you can select in the **Items** pane. To display the hook-ups in the **Items** pane, in the tree view pane, select the parent hook-up type or the **Hook-Ups** folder. When duplicating a hook-up, the software also duplicates all of the hook-up items that are associated with the source hook-ups.

Associate Hook-Up Types with Instrument Types

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. In the tree view pane, do one of the following:
 - Select and right-click a specific hook-up type.
 - Select the **Hook-Ups** folder and then, in the **Items** pane, select and right-click one or more hook-up types.
3. On the shortcut menu, click **Actions > Associate with Instrument Types**.

TIPS

- On the dialog box that opens, the software displays all the instrument types defined in the current domain. You can associate the same instrument type with more than one hook-up type.
 - In the dialog box panes, you can double-click a column header in the to sort the items in ascending order. Double-click again to sort the items in descending order.
4. Under **Available instrument types**, select one or more instrument types and do one of the following:
 - Click **Associate**.
 - Drag the selected items to the **Associated instrument types** pane.
 5. If you selected more than one hook-up type in the **Domain Explorer**, click **Next** to display another hook-up type and associate instrument types as you require.

Associate Instrument Tag Numbers with Hook-Ups

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a hook-up and then, In the **Items** pane, select and right-click one or more hook-ups.
 - In the tree view pane, select the hook-up hierarchy root node to display all the existing hook-up types in the **Items** pane, and then, in the **Items** pane, right-click one or more hook-up types.

TIP

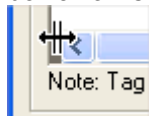
When selecting a hook-up type or several hook-up types, you can associate tags with multiple hook-ups that belong to different hook-up types. If you select a specific hook-up, you can only assign tag numbers to the selected hook-up.

3. On the shortcut menu, click **Actions > Associate Tag Numbers with Hook-Ups**.
4. Under **Instrument type assigned to hook-up type**, select an instrument type by which you want to filter tag numbers in the **Available tag numbers** pane.

TIPS

- In the dialog box panes, you can double-click a column header to sort the items in ascending order. Double-click again to sort the items in descending order.

- In the dialog box panes, there is a vertical bar that you can drag to the right and view your data in a split display. To find the vertical bar, place the cursor to the left of the left scroll box arrow so that the cursor changes its shape as shown:



- Under **Available tag numbers**, tags displayed in italics indicate tags that are already assigned to another hook-up. You can still associate these tags with the current hook-up. Click the **Hook-Ups** button above the **Available tag numbers** pane to display tag numbers available for association with hook-ups and also display existing hook-up associations.
 - If the specified hook-up is assigned to a user-defined sub-library with pipe specs, the software filters the tag numbers so that only tag numbers with the appropriate pipe spec data are displayed. The software only displays tag numbers whose pipe spec data fits the pipe specs assigned to the sub-library to which the hook-up belongs.
 - You can display the available tag numbers either on the highest or lowest level of your plant hierarchy defined by the Domain Administrator. The default highest level is Plant. The default lowest level is Unit.
5. Under **Available tag numbers**, select one or more tags and do one of the following:
 - Click **Associate**.
 - Drag the selected tags to the **Associated tag numbers** pane.
 6. If you selected more than one hook-up or hook-up type in the Domain Explorer, click **Next** and make associations for another hook-up.

TIP In the **Associated tag numbers** pane you can select to include specific tags in a **Bill of Material (BOM)**, or not to display the tag the next time you open the dialog box, or when you generate a CAD or Enhanced Utility Report Hook-Up drawing.

Move Hook-Ups to Another Hook-Up Type

1. Press F7 to open the **Domain Explorer**, and double-click the **Hook-Ups** folder to display all the existing hook-ups types and hook-ups in the **Items** pane.
2. In the **Items** pane, select and right-click hook-ups that you want to move to another hook-up type.
3. On the shortcut menu, click **Actions > Move Hook-Ups**.
4. On the dialog box that opens, select the target hook-up type and click **OK**.

Change a Hook-Ups Sub-Library Association

1. Press F7 to open the **Domain Explorer**, and double-click the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, right-click a specific hook-up.
 - In the **Items** pane, select and right-click one or more hook-ups.
3. On the shortcut menu, click **Properties**.

4. On the **Hook-Up Properties** dialog box, from the **Item sub-library** list, select a desired sub-library.

TIPS

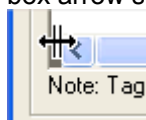
- The **Item sub-library** contains all sub-libraries you defined in the active item library in the **Reference Explorer**, where the active library is indicated with the  icon. If you set another library as the active item library, the software automatically assigns the default sub-library of the new active item library to all the existing hook-ups.
- If you assigned the hook-up to one of the user-defined sub-libraries, you can click **Pipe Specs** to display the pipe specs you associated with the selected user-defined sub-library in the **Reference Explorer**.

Associate Items with Hook-Ups

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a hook-up type and then, in the **Items** pane, select and right-click one or more hook-ups.
 - In the tree view pane, select the **Hook-Ups** folder to display all the existing hook-up types and hook-ups in the **Items** pane, and then, in the **Items** pane, right-click one or more hook-up types or hook-ups.
3. On the shortcut menu, click **Actions > Associate Items with hook-Ups**.

TIPS

- On the dialog box that opens, the software displays all the items available in the library you set in the **Reference Explorer** as the active item library.
- In the dialog box panes, you can double-click a column header in the to sort the items in ascending order. Double-click again to sort the items in descending order.
- In the dialog box panes, there is a vertical bar that you can drag to the right and view your data in a split display. To find the vertical bar, place the cursor to the left of the left scroll box arrow so that the cursor changes its shape as shown:



4. Under **Items available for association**, select one or more hook-up items and do one of the following:
 - Click **Associate**.
 - Drag the selected items to the **Associated items** pane.
5. In the **Quantity** column, type a value for the quantity of items for a Bill of Material. For example, if you associated a male connector and the current hook-up requires twenty male connectors, type **20**.
6. In the **Spares Percentage** column, type or modify the percentage of total units of measure specified for the current hook-up item on the **Hook-Up Item Properties** dialog box.

TIPS

- The software uses the values that you add in the **Quantity** and **Spares Percentage** columns when calculating the total quantity of items for a Bill of Material. For details, see *Calculating the Total of Hook-Up Items for a Bill of Material* (on page 346).

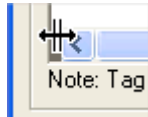
- After associating hook-up items with a hook-up, the software generates new sequence numbers automatically and displays the numbers in the **Order** column, where you can sort the items by their order of their assignment to a hook-up. You can type a new sequence number if needed. You can open a Bill Of Material and sort the items by the item order. For more information, see *Sort Data in a Bill of Material* (on page 348).

Dissociate Items from Hook-Ups

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a hook-up type and then, in the **Items** pane, select and right-click one or more hook-ups.
 - In the tree view pane, select the **Hook-Ups** folder to display all the existing hook-up types and hook-ups in the **Items** pane, and then, in the **Items** pane, right-click one or more hook-up types or hook-ups.
3. On the shortcut menu, click **Actions > Associate Items with hook-Ups**.

TIPS

- On the dialog box that opens, under **Associated items**, you can double-click a column header in the to sort the items in ascending order. Double-click again to sort the items in descending order.
- In the **Associated items** pane, there is a vertical bar that you can drag to the right and view your data in a split display. To find the vertical bar, place the cursor to the left of the left scroll box arrow so that the cursor changes its shape as shown:



4. Under **Associated items**, select one or more hook-up items and do one of the following:
 - Click **Dissociate**.
 - Drag the selected items from the **Associated items** pane to the **Items available for association** pane.

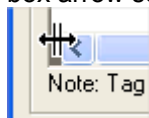
Dissociate Tag Numbers from Hook-Ups

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a hook-up and then, In the **Items** pane, select and right-click one or more hook-ups.
 - In the tree view pane, select the hook-up hierarchy root node to display all the existing hook-up types in the **Items** pane, and then, in the **Items** pane, right-click one or more hook-up types.
3. On the shortcut menu, click **Actions > Associate Tag Numbers with Hook-Ups**.

TIPS

- In the dialog box panes, you can double-click a column header in the to sort the items in ascending order. Double-click again to sort the items in descending order.

- In the dialog box panes, there is a vertical bar that you can drag to the right and view your data in a split display. To find the vertical bar, place the cursor to the left of the left scroll box arrow so that the cursor changes its shape as shown:



4. Under **Associated tag numbers**, select one or more tags and do one of the following:
 - Click **Associate**.
 - Drag the selected tags to the **Available tag numbers** pane.
5. If you selected more than one hook-up or hook-up type in the **Domain Explorer**, click **Next** and dissociate tag numbers from another hook-up.

Delete Hook-Ups

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a specific hook-up you want to delete.
 - In the tree view pane, select a hook-up type, and then, in the **Items** pane, select and right-click one or more hook-ups.
3. On the shortcut menu, click **Delete**.

TIP When deleting hook-ups, the software automatically dissociates all the hook-up items and instrument tags.

Delete Hook-Up Types

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, right-click a hook-up type that does not have hook-ups.
 - In the **Items** pane, select and right-click several hook-up types that do not have hook-ups.
3. On the shortcut menu, click **Delete**.

Hook-Up Drawing Generation

You can generate a hook-up drawing using either the Enhanced Report Utility or an external CAD application such as SmartSketch, AutoCAD, or MicroStation. Generated drawings contain the hook-up block drawing, the list of assigned tag numbers, and hook-up items associated with the current hook-up.

CAD Hook-Up Drawing Generation Scenario

1. On the **Preferences** dialog box, on the **Hook-Ups > File Locations** page, define the CAD paths for hook-up drawings.
2. If you require a global border and logo, in your CAD application, prepare a drawing border and logo, and then, enable the use of it in SmartPlant Instrumentation. For details, see *Assign Border and Logo Blocks to Hook-Up Drawings* (on page 337).

3. In your CAD application, create a general drawing (optional). You only need to create such a drawing if you intend to use the **General drawing** option of SmartPlant Instrumentation when generating a hook-up drawing that contains more than one page. Then, on the SmartPlant Instrumentation **Preferences** dialog box, on the **Hook-Ups > General** page, specify the general drawing that you want to appear on the second and all other subsequent pages of the hook-up drawing you will generate.

TIP A generated hook-up drawing, using the **General drawings** options, contains more than one page. Your general drawing will appear on the second page and all other subsequent pages of the hook-up drawing generated from SmartPlant Instrumentation. All the tags that appear on the first page are duplicated on the second page along with those tags that do not fit on the first page.

4. On the **Preferences** dialog box, on the **Hook-Ups > General** page, set the revision display order.

TIP You can display a number of revisions in the title block of a hook-up drawing. Each revision is represented by a numbered macro and the revisions are displayed in the order of the macro numbering, beginning with the first revision (ascending order) or the last revision (descending order). If the total number of revisions is greater than the number of lines available for display in the drawing and you want to see the latest revisions, you should select to display the revisions in descending order so that the latest revision is displayed first.

5. In your CAD application, create a hook-up drawing template using your CAD application so that this template includes the tag number and hook-up item macros as well as the title block.

IMPORTANT Make sure the macro names you enter in the drawing template match the macro names you defined on the **Macro Definitions** dialog box in the Hook-Ups module.

6. If you want to edit the hook-up drawing identifying fields, associate the document and revision numbers, in the **Domain Explorer**, open the **Hook-Up Drawing List** dialog box and define data accordingly.
7. In the **Domain Explorer**, open the **Generate Hook-Up Drawings** dialog box and set the generation options as you require.
8. Generate a hook-up drawing from the **Generate Hook-Up Drawings** dialog box.

Enhanced Hook-Up Drawing Generation Scenario

1. On the **Preferences** dialog box, on the **Enhanced Reports > Hook-Up > View** page, select the options you want to see in your drawing.
2. On the **Enhanced Reports > Hook-Up > File Locations** page, set the file paths for the various options.
3. If required, change the text appearance by changing the settings on the **Fonts** page.
4. On the **Enhanced Reports > Hook-Up > Title Block** page, select the title block you require for your drawing.
5. On the **Enhanced Reports > Hook-Up > Page and Revisions** page, set the revision display order.

TIP You can display a number of revisions in the title block of a hook-up drawing. Each revision is represented by a numbered macro and the revisions are displayed in the order of the macro numbering, beginning with the first revision (ascending order) or the last revision (descending order). If the total number of revisions is greater than the number of lines available for display in the drawing and you want to see the latest revisions, you should select to display the revisions in descending order so that the latest revision is displayed first.

6. If you want to edit the hook-up drawing identifying fields, associate the document and revision numbers, in the **Domain Explorer**, open the **Hook-Up Drawing List** dialog box and define data accordingly.

7. Assign the required hook-up symbol file to the hook-up. For more information, see *Assign an Enhanced Symbol to a Hook-Up* (on page 338).
8. In the **Domain Explorer**, open the **Generate Hook-Up Drawings** dialog box and set the generation options as you require.
9. Generate a hook-up drawing from the **Generate Hook-Up Drawings** dialog box.

Shipped Enhanced Hook-Ups Symbols

SmartPlant Instrumentation supplies two symbols for use with enhanced hook-ups;

- HookUp_CV.sym
- HookUp_DPtransmit.sym

They are located in the SmartPlant Instrumentation home folder\RAD\Symbols\Custom.

These shipped files can be used, as is, to produce enhanced hook-ups drawings for control valves, or data process transmitters. They can also be customized to produce enhanced hook-up drawings for other hook-up items. For more information on editing and customizing symbols, see the Symbol Editor or Catalog Manager User Guides.

See Also

Assign an Enhanced Symbol to a Hook-Up (on page 338)

Managing Hook-Up Drawings Common Tasks

These tasks deal with generating hook-up drawings using a CAD application or the Enhanced Report Utility, maintaining revisions, editing hook-up drawing identifying fields that appear in the title block, and so forth. For a complete list of hook-up reports, see *Hook-Up Reports* (on page 345).

Apply a Drawing Generation Method to Hook-Ups

Use this procedure to apply a drawing generation method to one or more hook-ups manually. This feature is useful if you intend to generate hook-up drawings in a mixed mode, that is, generate several CAD hook-up drawings and several enhanced hook-up drawings in one generation session. You generate CAD hook-up drawings using your CAD application (SmartSketch, AutoCAD, or MicroStation), whose paths you specified on the **Preferences** dialog box. You generate enhanced hook-up drawings using the Enhanced Report Utility, which comes with the SmartPlant Instrumentation software package. For more information, see *Apply a Drawing Generation Method to Hook-Ups* (on page 335).

Set Default Generation Method for Hook-Up Drawings

Use this procedure to set the default generation method that the software can use automatically when your hook-up selection contains hook-ups that do not have a generation method applied manually. For more information, see *Set Default Generation Method for Hook-Up Drawings* (on page 335).

Edit Hook-Up Drawing Identifying Fields

Hook-up drawing properties appear in the title block fields. These fields contain identifying information about the drawing name, page number, and file name, and also information about revisions, approvals, and references to other documents. You can modify the property information using the options in the **Hook-Up Drawing List** dialog box.

For each hook-up drawing, SmartPlant Instrumentation displays the hook-up name and hook-up description. You can use this procedure to edit the hook-up name, description, and the output file name. For more information, see *Edit Hook-Up Drawing Identifying Fields* (on page 335).

Make Approvals for Hook-Up Drawings

This procedure enables you to make client and vendor approvals for hook-up drawings. For more information, see *Make Approvals for Hook-Up Drawings* (on page 336).

Maintain Hook-Up Drawing Revisions

The revision feature is used to keep track of the changes made to the hook-up drawing during its lifetime. It is important and useful to have a chronological description of the changes, dates of change, and a list of persons who approved them. Using this procedure, you can add, edit, and delete revisions.

The revision feature is used to keep track of the changes made to the hook-up drawing during its lifetime. It is important and useful to have a chronological description of the changes, dates of change, and a list of persons who approved them. Using this procedure, you can add, edit, and delete revisions. For more information, see *Maintain Hook-Up Drawing Revisions* (see "Maintain Hook-Up Drawing Revisions" on page 336).

Add Document References in Hook-Up Drawings

This procedure enables you to make a reference to a drawing and add this reference to the title block of the hook-up drawing. For more information, see *Add Document References in Hook-Up Drawings* (on page 337).

Assign Border and Logo Blocks to Hook-Up Drawings

This procedure allows you to set SmartPlant Instrumentation to use a specific border block and logo block every time you generate a CAD hook-up drawing. Blocks that you set as default general blocks must belong to the shipped general block types, that is, Border and Logo block types available in the **Drawing Block Types** folder of the **Domain Explorer**. For more information, see *Assign Border and Logo Blocks to Hook-Up Drawings* (on page 337).

Assign a Hook-Up Symbol to a Hook-Up

Use this procedure to assign a hook-up symbol to your hook-up when generating an enhanced hook-up drawing using the Enhanced Report Utility. The hook-up symbol is of the file type .sym. For more information, see *Assign an Enhanced Symbol to a Hook-Up* (on page 338).

Update Paths for Hook-Up Drawings and Block Files

Use this procedure to update in the database paths for existing hook-up drawings and block files. You can update paths globally for block files associated with hook-ups, for logo and border blocks, and for hook-up drawings that you generated. For more information, see *Update Paths for Hook-Up Drawings and Block Files* (on page 338).

Generate CAD Hook-Up Drawings

Use this procedure to generate one or more hook-up drawings using your CAD application. For more information, see *Generate CAD Hook-Up Drawings* (on page 339).

Generate Enhanced Hook-Up Drawings

Use this procedure to generate one or more hook-up drawings using the Enhanced Report Utility. For more information, see *Generate Enhanced Hook-Up Drawings* (on page 340).

Generate Hook-Up Drawings in Mixed Mode

Use this procedure to generate several hook-up drawings according to the generation method applied to the source hook-ups. For more information, see *Generate Hook-Up Drawings in Mixed Mode* (on page 341).

Apply a Drawing Generation Method to Hook-Ups

1. Press F7 to open the **Domain Explorer** and then double-click the **Hook-Ups** folder.
2. In the **Items** pane, select and right-click one or more hook-ups.
3. On the shortcut menu, click **Apply Generation Method**.
4. Select one of the following:
 - **CAD** — Allows you to and assigns the indicator to the selected hook-ups.
 - **Enhanced Report** — Allows you to and assigns the indicator to the selected hook-ups.

TIPS

- To view an existing generation method assignment, you can right-click the appropriate hook-up and click **Properties** to open the **Hook-Up Properties** dialog box.
- To clear the current generation method for one or more hook-ups, select and right-click the required hook-ups, and then, on the shortcut menu, click **Actions > Clear Generation Method**.

Set Default Generation Method for Hook-Up Drawings

1. Click **File > Preferences > Hook-Ups > General**.
2. From the **Default generation method** list, one of the following methods:
 - **CAD** — Generates hook-up drawings using your CAD application and allows you to use the **CAD Options** tab.
 - **Enhanced Report** — Generates hook-up drawings using the Enhanced Report Utility.

TIP The default generation method that you set appears on the General Tab (Generate Hook-Up Drawings Dialog Box).

Edit Hook-Up Drawing Identifying Fields

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a hook-up type and then, in the **Items** pane, select and right-click one or more hook-ups.
 - In the tree view pane, select the **Hook-Ups** folder to display all the existing hook-up types and hook-ups in the **Items** pane, and then, in the **Items** pane, right-click one or more hook-up types or hook-ups.
3. On the shortcut menu, click **Actions > Hook-Up Drawing List**.
4. On the **Hook-Up Drawing List** dialog box, select a row and click **Edit**.
5. In the **Document** field, type a name for the drawing document.

TIP If you type an existing document name, the software prompts you to confirm the multiple use of the name.

6. In the **Document Description** field, type a description.
7. To change the default output file name, in the **Output File Name** field, type a different file name over the default name.

TIP The default output file name is the same as the hook-up name. If there are any spaces in the hook-up name, the software replaces the space with an underscore (_). The file name ends with an underscore (_) and a two-digit sequence number representing the page number of a multi-page drawing. For example, flow_transmitter_03.dwg. When the drawing consists of a single page, the file name ends with _01.

8. Repeat the appropriate steps of this procedure for any other hook-up drawing.

Make Approvals for Hook-Up Drawings

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a hook-up type and then, In the **Items** pane, select and right-click one or more hook-ups.
 - In the tree view pane, select the **Hook-Ups** folder to display all the existing hook-up types and hook-ups in the **Items** pane, and then, in the **Items** pane, right-click one or more hook-up types or hook-ups.
3. On the shortcut menu, click **Actions > Hook-Up Drawing List**.
4. On the **Hook-Up Drawing List** dialog box, click **Approvals** and then enter the client and vendor data as needed.

Maintain Hook-Up Drawing Revisions

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a hook-up type and then, In the **Items** pane, select and right-click one or more hook-ups.
 - In the tree view pane, select the **Hook-Ups** folder to display all the existing hook-up types and hook-ups in the **Items** pane, and then, in the **Items** pane, right-click one or more hook-up types or hook-ups.
3. On the shortcut menu, click **Actions > Hook-Up Drawing List**.
4. On the **Hook-Up Drawing List** dialog box, select a desired row and click **Revisions**.
5. On the **Revisions** dialog box, select one of the revision numbering methods (use **P0**, **P1**, **P2...** for preliminary revisions or **0**, **1**, **2** / **A**, **B**, **C**, and so forth, for normal serial revisions).

TIP When you first select a revision numbering method, several options are available to you, including preliminary revisions (designated by **P0**, **P1**, **P2**). Once you select one of the other revision methods, you will not be able to return to the preliminary revision method and this option will be disabled.

6. Do one of the following:
 - Click a field of an existing row to update the existing revision data.

- Click **New** to add new revision data, and then type new revision data values.

TIP As a time saver and a forget-me-not precaution, take advantage of using a default revision method. The software automatically adds a new line with the next logical character and date each time you click **New** after you select the initial method.

7. If necessary, click **Delete** to delete any previous revisions that are no longer needed.

TIP You can also maintain revisions in batch mode using global revisions. For more information, see Global Revisions.

Add Document References in Hook-Up Drawings

1. Press F7 to open the **Domain Explorer**, and expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. Do one of the following:
 - In the tree view pane, select a hook-up type and then, in the **Items** pane, select and right-click one or more hook-ups.
 - In the tree view pane, select the **Hook-Ups** folder to display all the existing hook-up types and hook-ups in the **Items** pane, and then, in the **Items** pane, right-click one or more hook-up types or hook-ups.
3. On the shortcut menu, click **Actions > Hook-Up Drawing List**.
4. On the **Hook-Up Drawing List** dialog box, click **References** to open the **Document References** dialog box.
5. To insert a new document reference, do the following:
6. Click **New** to open the **Drawing Reference Properties** dialog box.
7. Select a value from the **References** list.
8. Type the name and description in the appropriate boxes.
9. Click **OK** to return to the **Document References** dialog box.
10. To edit a manually inserted drawing reference, do the following:
11. Select a desired row and click **Properties** to open the **Drawing Reference Properties** dialog box.
12. Edit the name and description.
13. Click **OK** to return to the **Document References** dialog box.

TIP To delete a reference, select a row, and click **Delete**.

Assign Border and Logo Blocks to Hook-Up Drawings

1. In the **Domain Explorer**, expand the hierarchy of the **Drawing Block Types** folder.
 2. Right-click the **Border** block type, and then, on the shortcut menu, click **New > Block**.
 3. Define the new border block as you require.
 4. Right-click the **Logo** block type, and then, on the shortcut menu, click **New > Block**.
 5. Define the new logo block as you require.
- TIP** In hook-up drawings, you can only use the blocks created in default **Border** and **Logo** general block types. Blocks belonging to user-defined general block types, which you create for loop drawings, do not appear in hook-up drawings.
6. Click **File > Preferences > Hook-Ups > General**.
 7. Select **Use global border/logo**.

8. In the Hook-Ups module menu bar, click **Associations > Default General Blocks**.
9. On the **DefaultGeneral Blocks** dialog box, select the border and logo blocks from the lists.

Assign an Enhanced Symbol to a Hook-Up

1. In the **Domain Explorer**, open the **Hook-Up** folder
2. Right-click the required hook-up, and from the shortcut menu click **Properties**.
3. On the **Hook-Up Properties** dialog box, locate the **Drawing block file name and path** list, and click **Browse**.
4. On the **Select File** dialog box, navigate to, and select the .sym file you want to associate with the hook-up.

TIP If you want to view the symbol before assigning it to the hook-up, on the **Hook-Up Properties** dialog box, click **View** to display the symbol in your default symbol editor.

5. Click **OK**.

Update Paths for Hook-Up Drawings and Block Files

NOTE The paths that you specify do not overwrite the settings specified for new block files and output drawings on the **Hook-Ups > File Locations** page of the **Preferences** dialog box.

Open the Hook-Ups module.

1. Click **Actions > Update Paths for Existing Drawings and Blocks**.
2. Under **Path for associated block files**, do one of the following:
 - Click **Apply to drawings in the domain or project** to apply the new path to all the <plants> in the current domain, or to the current project in an owner operator domain.
 - Click **Apply to drawings in the current <plant> only** to apply the new path to the current <plant> only.
3. Type the full path to the folder where you keep your drawing blocks files or click **Browse** to navigate to the desired folder.
4. Under, **Path for generated hook-up drawings** do one of the following:
 - Click **Apply to blocks in the domain or project** to apply the new path to all the <plants> in the current domain, or to the current project in an owner operator domain.
 - Click **Apply to blocks in the current <plant> only** to apply the new path to the current <plant> only.
5. Type the full path to the folder where you keep your generated hook-up drawings or click **Browse** to navigate to the desired folder.

Generate CAD Hook-Up Drawings

NOTE A complete hook-up drawing generation scenario is described in the Help topic *Hook-Up Drawing Generation* (on page 331).

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. In the tree view pane, select a hook-up type.
3. In the **Items** pane, select and right-click one or more hook-ups.

TIP In the **Domain Explorer**, you can also select hook-up types instead of hook-ups. In this case, you can generate hook-up drawings that belong to all the hook-ups that are assigned to the selected hook-up types.

4. On the shortcut menu, click **Reports > Generate Hook-Up Drawings** to display the **General** tab of the **Generate Hook-Up Drawings** dialog box.
5. From the **Generation method** list, select **CAD**.
6. Click the **CAD Options** tab.
7. If you want to save the generated hook-up drawing as external CAD files, select the **Save file in format** check box, and then beside the **Target folder path** box click **Browse** to specify the folder. To select a folder, you have to select any file that exists in that folder.

TIPS

- In the **Target folder path** box, the software displays the default folder you specified on the **Preferences** dialog box, in the **Hook-Ups > CAD File Locations** page, in the **Output drawing folder** box.
 - The file format that the software uses when saving a hook-up drawing depends on your CAD application. If you use AutoCAD, the software saves the drawing file in .dwg format. If you select MicroStation, the software saves the drawing file in .dgn format.
 - When using SmartSketch, you can save the drawings in any format available in the list. From the box below **Save file in format**, select a desired format. If you select **SmartSketch**, the software saves the drawing file in .sym format.
 - The output filename is the same as the hook-up name. If there are any spaces in the hook-up name, the software replaces the space with an underscore (_). The filename ends with an underscore (_) and a two-digit sequence number representing the page number of a multi- page drawing. For example, flow_transmitter_03.dwg.
8. If needed, select **Send to plotter or printer** to send the generated drawings to the current CAD plotter or printer.
 9. If you want to apply the macro functions associated with macros, select **Use macro functions**.
 10. For a multi-page hook-up drawing generation, under **Drawing for subsequent pages**, select one of the following:
 - **Same as first** — Allows you to display the generated hook-up drawing on the first page as well as on all the subsequent pages as well.

- **General drawing** — Allows you to display the generated hook-up drawing on the first page and the general drawing on all the subsequent pages. The general drawing is a predefined drawing that you specified on the **Preferences** dialog box.

TIPS

- The software generates a multi-page drawing automatically. If the number of tag numbers associated with the selected hook-up is greater than the number of macros that can be accommodated on the first hook-up drawing page, then all the tag numbers appear on the second page (including those that already appeared on the first page). For a multi-page drawing, you can enter the page number of the first generated page of the hook-up drawing in the **Generate from page** box. For example, if you enter the value **2** and then generate a hook-up drawing, the first page of the hook-up drawing displays the value **2**. A multi-page drawing is a drawing in which the number of tags associated with the current hook-up is larger than the number of macros that can be accommodated in the generated hook-up drawing.
- If your CAD application is SmartSketch, by default, all drawings that you generate have the A3 Wide print sheet size. This print sheet size is defined in the template file pid.igr, which comes shipped with SmartPlant Instrumentation. This file appears in the CADFunc folder, for which you specified the path setting on the **Preferences** dialog box, **Hook-Ups > CAD File Locations** page. If you want to generate hook-up drawings using a different print sheet size, you must open the pid.igr file in SmartSketch and modify the sheet settings as you require. Note, however, that changes that you make in the pid.igr file affect both hook-up drawings and loop drawings you generate in SmartSketch. If you want the print sheet size of hook-up drawings not to affect the print sheet size of loop drawings, duplicate the CADFunc folder with all its content and define a different path setting preference for loop drawings on the **Loop Drawings > CAD File Locations** page.
- SmartPlant Instrumentation and the CAD drawing programs required to work with SmartPlant Instrumentation, must be installed on the same machine.

Generate Enhanced Hook-Up Drawings

NOTE A complete hook-up drawing generation scenario is described in the Help topic *Hook-Up Drawing Generation* (on page 331).

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. In the tree view pane, select a hook-up type.
3. In the **Items** pane, select and right-click one or more hook-ups.

TIP In the **Domain Explorer**, you can also select hook-up types instead of hook-ups. In this case, you can generate hook-up drawings that belong to all the hook-ups that are assigned to the selected hook-up types.

4. Assign the required hook-up symbol file to the hook-up. For details, see *Assign an Enhanced Symbol to a Hook-Up* (on page 338).
5. On the shortcut menu, click **Reports > Generate Hook-Up Drawings** to display the **General** tab of the **Generate Hook-Up Drawings** dialog box.
6. From the **Generation method** list, select **Enhanced Report**.

TIP Under **Automatic save options for enhanced hook-up drawings**, you can select **Save drawings automatically** to overwrite the default folder specified on the **Preferences** dialog box, **Enhanced Report > Hook-Ups > File Locations** page. You can click **Browse** and specify a different folder for saving the hook-up drawings automatically as files at the time of generation.

7. Click **OK** to generate the drawings for the selected hook-ups.

NOTE Generating an enhanced hook-up drawing with a **Tag List** and/or an **Associated Item List**, results in the lists being randomly positioned on your drawing. You can drag the lists and position them on the drawing as required. Each time you regenerate the drawing, the lists are randomly repositioned. To save the positions of the lists, drawing items, and so forth, in a Hook-Up drawing, use the **File > Save Custom Changes > Save position at > Drawing level** or **Layout level** command in the Enhanced Report Utility. For more information, see the **Using Enhanced Report Layouts** and the **Saving Custom Changes to Enhanced Reports** sections in the Enhanced Report Utility online help.

Generate Hook-Up Drawings in Mixed Mode

NOTES

- When generating hook-up drawings in mixed mode, you use the **As previously applied** option of the **Generate Hook-Up Drawings** dialog box. Mixed mode enables you to generate CAD hook-ups drawings and enhanced hook-up drawings altogether: CAD hook-up drawings for hook-ups whose generation method in the **Domain Explorer** is indicated as **C** and enhanced hook-up drawings for hook-ups whose generation method in the **Domain Explorer** is indicated as **E**. You must first assign your hook-ups to the appropriate generation methods. For details, see *Apply a Drawing Generation Method to Hook-Ups* (on page 335).
 - A complete hook-up drawing generation scenario is described in the Help topic *Hook-Up Drawing Generation* (on page 331).
1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
 2. In the tree view pane, select a hook-up type.
 3. In the **Items** pane, select and right-click one or more hook-ups.

TIPS

- In the **Domain Explorer**, you can also select hook-up types instead of hook-ups. In this case, you can generate hook-up drawings that belong to all the hook-ups that are assigned to the selected hook-up types.
 - For hook-up drawing generation in mixed mode, you selection can include hook-ups assigned to the **CAD** generation method, hook-ups assigned to the **Enhanced Reports** method, and also hook-ups that are not assigned to any generation method.
4. On the shortcut menu, click **Reports > Generate Hook-Up Drawings** to display the **General** tab of the **Generate Hook-Up Drawings** dialog box.
 5. Click the **CAD Options** tab and define options for CAD hook-up drawings as you require.
- TIP** You must first set generation options for CAD hook-up drawings because after you select **As previously applied** on the **General** tab, the **CAD Options** tab becomes disabled. For example, on the **CAD Options** tab, you can select the **Send to plotter or printer** check box if you want to print out the generated CAD hook-up drawings at the time of generation. This check box is cleared by default.
6. Click the **General** tab.
 7. From the **Generation method** list, select **As previously applied**.

TIPS

- After you select **As previously applied**, the **CAD Options** tab and the **Automatic save options for enhanced hook-up drawings** become disabled. When generating the drawings, the software saves the output files to the default folders whose paths you specified on the **Preferences** dialog box individually for CAD drawings and enhanced hook-up drawings, which are generated by the Enhanced Report Utility.

- If you hook-up selection in the **Domain Explorer** includes hook-ups that are not assigned to any method, the software applies the method displayed in the **Default generation method** box. You define the default generation method on the **Hook-Ups > General** page of the **Preferences** dialog box.
8. Click **OK** to generate the hook-up drawings for the selected hook-ups.

Displaying CAD Hook-Up Drawings with the Enhanced Report Utility Common Tasks

These tasks deal with using the Enhanced Report Utility to display hook-up drawings originating in AutoCAD, MicroStation and SmartSketch. Hook-up drawings comprise of a drawing, a title block, a block, and macros within the block. The Enhanced Report utility handles each CAD format differently.

IMPORTANT The Enhanced Report utility recognizes only tag and hook-up item macros within a given CAD block.

Display SmartSketch Hook-Up Drawings with the Enhanced Report Utility

Use this procedure to display a SmartSketch hook-up drawing in .sym format in the Enhanced Report Utility. You can display the drawing, block, and macros leaving the title block. For details, see *Display SmartSketch Hook-Up Drawings with the Enhanced Report Utility* (on page 342).

Display AutoCAD Hook-Up Drawings with the Enhanced Report Utility

Use this procedure to display an AutoCAD hook-up drawing in .dwg or .dxf format in the Enhanced Report Utility. You can display the block, drawing and the macros inside the block. However, you cannot associate an AutoCAD title block with the drawing. For details, see *Display AutoCAD Hook-Up Drawings with the Enhanced Report Utility* (on page 343).

Display MicroStation Hook-Up Drawings with the Enhanced Report Utility

Use this procedure to display a MicroStation drawing in .dgn format in the Enhanced Report Utility. You can display the drawing and the block leaving the block macros, and the title block. For details, see *Display MicroStation Hook-Up Drawings with the Enhanced Report Utility* (on page 343).

Display SmartSketch Hook-Up Drawings with the Enhanced Report Utility

1. Set the preferences to view the tag and item lists in the generated hook-up drawing. For details, see *Enhanced Reports > Hook-Ups > View (Preferences)*.
2. In SmartPlant Instrumentation, from the **Domain Explorer**, create a new hook-up item.
3. From the right click shortcut menu of the hook-up item, select **Properties**.
4. On the **Properties** dialog box, in the **Drawing block file name and path** data field, type the name and path of the desired SmartSketch block and click **Ok**.
5. Set the drawing generation method for this hook-up item to **Enhanced Report**. For details, see *Apply a Drawing Generation Method to Hook-Ups* (on page 335).
6. Generate the hook-up drawing. For details, see *Generate Enhanced Hook-Up Drawings* (on page 340).

IMPORTANT The Enhanced Report utility recognizes only tag and hook-up item macros within a SmartSketch block.

Display AutoCAD Hook-Up Drawings with the Enhanced Report Utility

IMPORTANT You must have AutoCAD open to be able to use this utility.

1. In the SmartPlant Instrumentation > RAD folder, double-click TranslateCADMacro.exe to start the utility that allows you to adjust the macros in an AutoCAD block to the Enhanced Report utility macro convention.
2. In the TranslateCADMacro utility, do the following:
 - a. In the **AutoCad blocks folder** pane, browse to and select the desired AutoCAD block folder.
 - b. In the **Target folder** pane, select the folder in which you want to place the adjusted blocks.
 - c. Click **OK**.
3. In SmartPlant Instrumentation, set the preferences to view the tag and item lists in the generated hook-up drawing. For details, see Enhanced Reports > Hook-Ups > View (Preferences).
4. From the **Domain Explorer**, create a new hook-up item.
5. Right click the Hook-up item, and on the shortcut menu, select **Properties**.
6. On the **Properties** dialog box, in the **Drawing block file name and path** data field, type the name and path of the desired AutoCAD block and click **OK**.
7. Set the drawing generation method for this hook-up item to **Enhanced Report**. For details, see *Apply a Drawing Generation Method to Hook-Ups* (on page 335).
8. Generate the hook-up drawing. For details, see *Generate Enhanced Hook-Up Drawings* (on page 340).

IMPORTANT The Enhanced Report utility recognizes only tag and hook-up item macros within an AutoCAD block.

Display MicroStation Hook-Up Drawings with the Enhanced Report Utility

1. Set the preferences to view the tag and item lists in the generated hook-up drawing. For details, see Enhanced Reports > Hook-Ups > View (Preferences).
2. In SmartPlant Instrumentation, from the **Domain Explorer**, create a new hook-up item.
3. From the right click shortcut menu of the hook-up item, select **Properties**.
4. On the **Properties** dialog box, in the **Drawing block file name and path** data field, type the name and path of the desired MicroStation block and click **OK**.
5. Set the drawing generation method for this hook-up item to **Enhanced Report**. For details, see *Apply a Drawing Generation Method to Hook-Ups* (on page 335).
6. Generate the hook-up drawing. For details, see *Generate Enhanced Hook-Up Drawings* (on page 340).

Create a New Hook-Up Macro Definition

1. From the Hook-Ups or Loop Drawings module, do one of the following:
 - Click **Tables > Macro Definitions**.
 - Click .
2. Select the desired macro by choosing the item type and property, and where applicable, the relation between a specified item type and its parent. To do this, make a selection from each of the lists at the top of the dialog box.
3. In the **Macro definition details** data window, select the desired macro definition.
TIP If there are a large number of macro definitions in the **Macro definition details** data window, under **Find macro definition**, select a column heading and then type a value. For example, if you select **Function** as a column heading and type a macro function value, in the data window, the software locates the macro definition to which that macro function belongs.
4. To create a customized macro definition, do one or more of the following:
 - a. Under **User-Defined Macro Name**, type the desired user name for your macro.
IMPORTANT The **User-Defined Macro Name** field must contain a name, it cannot be left empty. If a name is missing, the macro will not function and data could be lost.
 - b. Click in the **Description** field and change the description as desired.
 - c. Under **Macro Function**, select a macro function to modify the appearance of the retrieved data.
 - d. To add another customized macro definition, click **New** and repeat the previous step.
 - e. Click **Apply** to accept your customized macro definitions.

See Also

Using Macros in CAD Blocks (on page 295)

Delete a Hook-Up Macro Definition

1. From the Hook-Ups or Loop Drawings module, do one of the following:
 - Click **Tables > Macro Definitions**.
 - Click .
2. Select the desired macro by choosing the item type and property, and where applicable, the relation between a specified item type and its parent. To do this, make a selection from each of the lists at the top of the dialog box.
3. In the **Macro definition details** data window, select the row containing the macro definition that you want to delete.
TIP If there are a large number of macro definitions in the **Macro definition details** data window, under **Find macro definition**, select a column heading and then type a value. For example, if you select **Function** as a column heading and type a macro function value, in the data window, the software locates the macro definition to which that macro function belongs.
4. Click **Delete**.

See Also

Using Macros in CAD Blocks (on page 295)

Hook-Up Reports

The following hook-ups reports are available in SmartPlant Instrumentation:

- **Bill of Material** — Displays data used in a Bill of Material.
- **Hook-Up Tag List** — Displays instrument tag numbers assigned to hook-ups.
- **Hook-Up Item List** — Displays hook-up items assigned to hook-ups.
- **Library Items** — Displays items defined in the item libraries. If an item library contains user-defined sub-libraries, the items are displayed in the report per sub-library.
- **Hook-Up Macros** — Displays all SmartPlant Instrumentation macros used in hook-up drawings. The report displays the macro name, description, name used in SmartPlant Instrumentation and, if applicable, the macro function.

According to the report type, you can generate reports either using the **Reports** menu of either **Domain Explorer**, the **Reference Explorer**, or the **Hook-Ups** window. The following table lists the report-specific options:

Access to Report	Bill of Material	Hook- Up Tag List	Hook- Up Item List	Library Items	Hook- Up Macros
Domain Explorer	Allows you to select all or selected hook-up types, or hook-ups in the Hook-Ups folder.	Allows you to select all or selected hook-up types, or hook-ups in the Hook-Ups folder.	Allows you to select all or selected hook-up types, or hook-ups in the Hook-Ups folder.	N/A	N/A
Reference Explorer	N/A	N/A	N/A	Allows you to select all or specific libraries or sub-libraries in the Hook-Up Item Libraries folder	N/A

Access to Report	Bill of Material	Hook- Up Tag List	Hook- Up Item List	Library Items	Hook- Up Macros
Hook-Ups Module window	Allows you to open previously generated Bills of Materials Bill of Material by selecting the document number of the existing Bills of Materials.	Allows you to open previously generated Hook- Up Tag List reports by selecting the document number of the existing reports.	Allows you to open previously generated Hook- Up Item List reports by selecting the document number of the existing reports.	Allows you to click Reports > Library Items to generate a report of all existing hook-up items.	Allows you to click Reports > Hook-Up Macros and group the macros by module to organize the report pages according to the module titles. If you do not group the macros, the report displays all the macros in alphabetical order.

Calculating the Total of Hook-Up Items for a Bill of Material

When you generate a Bill of Material, SmartPlant Instrumentation calculates the total quantity of hook-up items according to the following formula: (number of associated instrument tags + number of additional tags for the Bill of Material) × quantity of items of the same type × (item spares percentage + 100) / 100. For example, if the total number of control valves associated with the hook-ups in the Bill of Material is 5, the number of additional instruments is 2, the quantity of male connector items is 2, and spares percentage is 20%, the total quantity is calculated as $(5 + 2) \times 2 \times (20 + 100) / 100$ and the Total Qty column in the Bill of Material displays 12.

NOTES

- If the unit of measure of the hook-up item is either **Item** or **Piece**, the software rounds up the calculated total quantity value so that no fraction remains.
- You define the number of the additional tags on the **Associate Tag Numbers with Hook-Ups** dialog box.
- You define the item quantity and spares percentage on the **Associate Items with Hook-Ups** dialog box. It is also possible to define item spares percentage on the **Hook-Up Item Properties** dialog box.

Hook-Up Reports Common Tasks

These tasks deal with generating and opening a print preview of a Bill of Material and reports that show hook-up data, the associated instruments, and hook-up items. For a complete list of hook-up reports, see *Hook-Up Reports* (on page 345).

Generate a Bill of Material

This procedure describes how to generate a Bill of Material (BOM), which is the main purpose of the Hook-Ups module. Prior to generating a Bill of Material, you need to define hook-up types and hook-ups in the Domain Explorer, and then, associate hook-up items and instruments to the hook-ups. For more information, see *Generate a Bill of Material* (on page 348).

Sort Data in a Bill of Material

Use this procedure to sort data in a Bill of Material print preview. For more information, see *Sort Data in a Bill of Material* (on page 348).

Generate a Hook-Up Item List Report

Use this procedure to generate a report that displays hook-up items associated with the hook-ups you select in the **Domain Explorer**. For more information, see *Generate a Hook-Up Item List Report* (on page 349).

Generate a Hook-Up Tag List Report

Use this procedure to generate a report that displays instrument tag numbers associated with the hook-ups you select in the **Domain Explorer**. For more information, see *Generate a Hook-Up Tag List Report* (on page 349).

Open Reports by Selecting Document Numbers

Use this procedure to open reports by selecting document numbers you created for the current report type. To use this procedure, you must first generate reports and define a document number for each report you want to make available for selection. For example, you can generate five Hook-Up Item List reports for hook-ups belonging to the **FLOW** hook-up type and define the **Flow Hook-Up** document number for each report. Furthermore, you can generate four Hook-Up Item List reports for hook-ups belonging to the **LEVEL** hook-up type and define the **Level Hook-Up** document number for each report. You can then use this procedure to select all or some of the defined document numbers and open the report print preview. For example, you can select five **Flow Hook-Up** document numbers and open the print preview of five Hook-Up Item List reports. For more information, see *Open Reports by Selecting Document Numbers* (on page 350).

Generate a Bill of Material

1. Press **F7** to open the **Domain Explorer** and then expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. In the tree view pane, select a hook-up type.
3. In the **Items** pane, select and right-click one or more hook-ups.
TIP In the **Domain Explorer**, you can also select hook-up types instead of hook-ups. In this case, you can generate a Bill of Material for all hook-ups that belong to the selected hook-up types. If you select all the hook-up types, you generate a Bill of Material for all the existing hook-ups.
4. On the shortcut menu, click **Reports > Bill of Material**.
5. Click **Yes** to open the print preview of the new Bill of Material or click **No** to send it to your default printer.

TIPS

- In the Bill of Material header area, some of the headers are not displayed if you generated a Bill of Material for more than one hook-up or hook-up type. For details, see Bill of Material Print Preview Window.
- In the **Print Preview** window, you can sort the Bill of Material data as you need and group the Bill of Material data by hook-up item manufacturer.
- If you select the **Edit mode** check box, you can enter values in the **For Order** field, which appears under the **Total Qty** field. The **For Order** values are not saved in the database and only appear in the current printout. For details about the calculation formula, see *Calculating the Total of Hook-Up Items for a Bill of Material* (on page 346).
- You can select **Print BOM together with the tag list** to print out the instrument tags associated with the hook-ups when printing the Bill of Material. The **Print Preview** window does not show the instrument tags.
- On the **Print Preview** window toolbar, clicking the revision icon  enables you to create the Bill of Material revision and define the document number. After you define the document number, you can then open the same report using the **Reports > Reports by Document Numbers** option of the main **Hook-Ups Module** window.
- Clicking  on the **Print Preview** window toolbar, allows you to save the Bill of Material to an external file and see the changes in the Bill of Material the next time you generate it.

Sort Data in a Bill of Material

1. In the Bill of Material print preview, from the **Sort by** list, select an option to arrange the fields as you require.
TIP Each option in the **Sort by** list corresponds to a column header in the print preview.
2. Select the **Ascending order** check box to display the fields in ascending order or clear the **Ascending order** check box to display the fields in descending order.

Generate a Hook-Up Item List Report

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. In the tree view pane, select a hook-up type.
3. In the **Items** pane, select and right-click one or more hook-ups.

TIP In the **Domain Explorer**, you can also select hook-up types instead of hook-ups. In this case, you can generate a report for all hook-ups that belong to the selected hook-up types. If you select all the hook-up types, you generate a report of all the hook-up items associated with for the existing hook-ups.

4. On the shortcut menu, click **Reports > Hook-Up Item List**.
5. Click **Yes** to open the report print preview or click **No** to send it to your default printer.

TIP On the **Print Preview** window toolbar, clicking the revision icon  enables you to create the report revision and define the document number. After you define the document number, you can then open the same report using the **Reports > Reports by Document Numbers** option of the main **Hook-Ups Module** window.

Generate a Hook-Up Tag List Report

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the hook-up type level in the **Hook-Ups** folder.
2. In the tree view pane, select a hook-up type.
3. In the **Items** pane, select and right-click one or more hook-ups.

TIP In the **Domain Explorer**, you can also select hook-up types instead of hook-ups. In this case, you can generate a report for all hook-ups that belong to the selected hook-up types. If you select all the hook-up types, you generate a report of all the tags associated with for the existing hook-ups.

4. On the shortcut menu, click **Reports > Hook-Up Tag List**.
5. Click **Yes** to open the report print preview or click **No** to send it to your default printer.

TIP On the **Print Preview** window toolbar, clicking the revision icon  enables you to create the report revision and define the document number. After you define the document number, you can then open the same report using the **Reports > Reports by Document Numbers** option of the main **Hook-Ups Module** window.

Open Reports by Selecting Document Numbers

NOTE To use this procedure, you must first generate the appropriate hook-up report in the **Domain Explorer**.

1. On the **Hook-Ups** module menu bar, click **Reports > Reports by Document Numbers**, and then, select the appropriate report.
2. On the dialog box that opens, select one or more document numbers and click **OK** to open the print preview of the reports.

Instrument Maintenance

The Maintenance module provides you with all the tools required to plan, carry out, and document the breakdown and preventive maintenance associated with the instruments in your plant.

Breakdown Maintenance

Breakdown maintenance features deal with the malfunction of equipment. You can deal with problems on two levels:

- **Work Requests**

A work request can be initiated by most of the technical staff. A work request describes a possible malfunction or repair that may be required in the future. The work requests are then examined by the maintenance supervisor or foreman who can approve them and then create an actual work order (repair).

- **Repair Forms**

A repair form is the actual repair or work that has been approved by the maintenance supervisor or foreman. It indicates the repair date and describes the nature of the problem as well as the work group that is assigned to carry out the job. The repair form also includes, in a different section, the information that is filled in by the person who performed the work, malfunction cause and action taken, as well as the down and repair time.

Preventive Maintenance

Preventive maintenance features enable you to maintain and schedule periodic **maintenance activities that contain tasks and procedures**.

NOTES

- The Maintenance module is available only if it is included in the software license that you purchased from Intergraph.
- You can issue new work requests, repairs, or preventive maintenance work orders only from the Instrument Index module. Once you have added maintenance records, you need to access the **Maintenance Activities Summary** window for further handling of these activities.

Access Rights in the Maintenance Module

The access rights in the Administration module that control the data input for the Maintenance module are summarized in the following table.

Level	Item	Description
<Plant>	Supporting tables	Ability to access, edit and delete items from the supporting tables of the module which controls the forms.
<Unit>	Module Access	Ability to allow or deny access to this module.
<Unit>	Work Request - Technician	Ability to add new work request Forms from the Index module. This option should be enabled to all technical staff.
<Unit>	Repair - Technician	Ability to convert and schedule the work request into an actual work order.
<Unit>	Repair - Supervisor	Technician's option includes filling in the repair data.
<Unit>	Preventive Maintenance - Supervisor	Preventive Maintenance supervisor's option to schedule PM activities, assign a work group and schedule dates.
<Unit>	Preventive Maintenance - Technician	Preventive Maintenance technician options include filling out test results.

Start the Maintenance Module

NOTE The Maintenance module is available only if it is included in the software license that you purchased from Intergraph. Before starting this module, check with the Domain Administrator to ensure that you have been granted appropriate access rights for the tasks you will carry out.

Starting the Maintenance Module From Any SmartPlant Instrumentation Module

- Do one of the following:
 - Click **Modules > Maintenance**.
 - On the main toolbar, click .

Starting the Maintenance Module From an Instrument Index Browser View

1. Open an Instrument Index Standard Browser view.
2. In the browser view, select and then right-click the tag numbers that require maintenance.
3. On the shortcut menu, point to **Maintenance** and then click one of the following options:
 - **Schedule Preventive Maintenance Activities**
 - **Work Request Form**
 - **Repair Form**

See Also

Performing Breakdown Maintenance (see "Performing Preventive Maintenance" on page 355)

Performing Preventive Maintenance (see "Performing Breakdown Maintenance" on page 359)

Filter Maintenance Activities

Use this procedure to filter the maintenance records of the current domain displayed in the **Maintenance Activities Summary** window. These include the following types of records:

- Preventive maintenance
 - Repair forms
 - Work requests
1. Open the **Maintenance Activities Summary** window.
 2. Under **Process function**, do one of the following:
 - To display records for tags of all process functions, select the blank row.
 - To limit the records to tags of one process function, select the process function of interest.
 3. To limit the display of preventive maintenance records to a given preventive maintenance code, select the code under **Preventive maintenance code**.
 4. To limit the display of maintenance records to a given date range, do the following:
 - a. Under **From**, enter the starting date.
 - b. Under **To**, enter the ending date.
 5. Do one of the following under **Data level**:
 - To limit the records displayed, select **Plant**, **Area**, or **Unit**.
 - To not limit the records displayed, accept the default setting of **Domain**.
 6. To limit the display of maintenance records by tag category, select a value from the **Category** list.
 7. In the **Maintenance records** pane, do one of the following:
 - To display records for all of the tags that have maintenance data, select **All Activities**.
 - To display records for selected tags, click  to open the tree, and then, select the tag for which you want to display maintenance records

TIP To clear all filter settings, click **Reset**.

Generate a Maintenance Report

This option enables you to select and generate a maintenance report. You can generate the following reports:

- Scheduled Preventive Maintenance Report
 - Summary of Failure Reason Report
 - Summary of Repair Action Report
 - Summary of Outcome Report
 - Repair Details Report
1. Start the Maintenance module.
 2. In the **Maintenance Activities Summary** window, on the **Reports** menu, click the report that you want to generate.
 3. On the **Report Properties** dialog box, in the **Date range** group box, do one of the following to set the date range for the report:
 - Select the **User defined** option and set a specific period by typing or selecting the required dates from the **From** and **To** lists.
 - Select the **Predefined** option to generate a report for the last, current or next week, month, or year.
 4. To filter the data level of the report, select **Plant**, **Area**, or **Unit**.

Preventive Maintenance Preliminary Activities

You have to define certain preliminary settings before you can start scheduling preventive maintenance activities or issuing work requests and repair forms. These preliminary settings include the following three definitions:

- *Define a Preventive Maintenance Attachment* (on page 353)
- *Define a Preventive Maintenance Task* (on page 354)
- *Define a Preventive Maintenance Activity* (on page 354)

These definitions then serve as references for the actual maintenance procedures carried out by the field maintenance personnel.

Define a Preventive Maintenance Attachment

An attachment is an external document that is provided to the maintenance crew. This procedure describes how to define an attachment and associate it with an external file, which can be a text document, a .doc file, a drawing, a sketch, and so forth.

1. Start the Maintenance module.
2. In the **Maintenance Activities Summary** window, do one of the following:
 - Click **Tables > Attachments**.
 - Click .
3. On the **Attachments** dialog box, click **New** to define a new attachment.
4. Click in the **Name** field and type the attachment name or code that will then serve as a reference for the task.
5. In the **Description** field, type an appropriate description.
6. Click **Browse** to associate the current attachment with an external file which can be a drawing, a text file, or a vendor's document, and so forth.

7. Click **View** to view the external file to make sure it is the correct one.
8. Click **Save** and then **Close** to return to the **Maintenance Activities Summary** window.

Define a Preventive Maintenance Task

After defining the preventive maintenance attachments, you have to define the appropriate maintenance tasks that will be carried out. The tasks are associated with the attachments you have defined and then they will be included in the appropriate maintenance activities.

1. Start the Maintenance module.
2. In the **Maintenance Activities Summary** window, do one of the following:
 - Click **Tables > Preventive Maintenance Tasks**.
 - Click .
3. On the **Preventive Maintenance Tasks** dialog box, click **New** to define a new task.
4. Click in the **Task** field and type the new task name.
5. Click in the **Attachment** field and select the appropriate attachment with which new task is to be associated.
6. Click in the **Note** field and type any note you deem fit for the current task.
7. Click **Save** and then **Close** to return to the **Maintenance Activities Summary** window.

Define a Preventive Maintenance Activity

You define the preventive maintenance activities after defining the preventive maintenance tasks and attachments. Each activity can hold a number of tasks which are in turn associated with the defined attachments. You select the defined tasks in the sequence they are to be performed. The defined activity also contains other default scheduling information, such as the down/repair time, the interval and frequency for the current PM activity, the required calibration values, and so forth.

1. Start the Maintenance module.
2. In the **Maintenance Activities Summary** window, do one of the following:
 - Click **Tables > Preventive Maintenance Activities**.
 - On the module toolbar, click .
3. On the **Preventive Maintenance Activities** dialog box, click **New** to define a new activity.
4. Click in the **PM code** field and type a new preventive maintenance code for the current activity.
5. Click in the **Typical Instrument** field and select the appropriate typical instrument from the list. If the required option is not available, click **Typical Instrument**.
6. Click in the **Description** field and type an appropriate description for the current activity.
7. Click in the **Note** field and type any appropriate note if necessary. Use the horizontal scroll bar to make the **Note** field visible.
8. In the **Associated preventive maintenance tasks** section, click **Add** to include a task with the activity.
9. Click in the **Task** field and select the appropriate task from the list. You can assign more than one task to the same activity. Click **Add** to add more tasks as required. Note that each task must be unique. If the required task is not available, click **Tasks**.

The software assigns the new task a sequence number. The appropriate attachment is selected and it appears in the **Attachment** field.

10. Enter the default scheduling data as needed by selecting the appropriate scheduling settings from the **Default scheduling** and **Estimated maintenance time** sections.

NOTE Selecting the **Calibration Required** check box enables you to enter calibration values when filling out the Preventive Maintenance details. This option becomes available in the **Preventive Maintenance Activity Details** dialog box if you have defined the required calibration settings for the current tag before you started the Maintenance module.

11. Click **Save** and then **Close** to return to the **Maintenance Activities Summary** window.

Performing Preventive Maintenance

Scheduling new preventive maintenance work orders is done only from the Instrument Index module (unless you are extending an already assigned PM job). Once a PM work order has been scheduled, you open the **Maintenance Activities Summary** window for further handling of these activities where you fill in the maintenance details or reports.

To define the raw PM activities, tasks and attachments, you need to access the Maintenance module and start to define the tasks from the lowest level – the 'attachment'. An 'attachment' describes a set of instructions to the technical staff. An attachment can be associated with any external document (a Microsoft Word file, a scanned image, or manufacturer maintenance document). Typical attachments define safety precaution instructions, technical instructions for bypassing a process, the actual set of instructions required to carry out the activity itself, restoring the process back to its operational status, and so forth.

A task is a set of instructions associated with an attachment. You can associate a single attachment with multiple PM activities. A task can include the attachment and a description.

Once you define the attachments and tasks, you can introduce a new PM activity. Each PM activity can hold several tasks in the sequence they were added. At this stage, it is also advisable to associate a PM code with a particular typical instrument (for example, transmitter, transducer, control valve, switch, and so forth.) This is very useful for report generation and other statistics. Moreover, a PM activity contains additional default information, such as the required calibration values, the down/repair time that an average maintenance job will take, the interval and frequency for each PM job.

Once all the required attachments, tasks, and activities have been defined, the preventive maintenance supervisor can schedule these activities to the selected tags from an Instrument Index Standard Browser view. At this stage, the PM activities are ready to be further processed and be filled in by the maintenance crew in due time.

Perform Preventive Maintenance

The following is the recommended flow of activities for preventive maintenance:

1. Start the Maintenance module and define the raw PM activities, tasks, and attachments.
2. Start the Instrument Index module and open an Instrument Index Standard Browser view.
3. Select the required tag numbers for which you want to schedule a PM job and start the Calibration module.
4. If the instrument requires calibration, enter the appropriate calibration settings for the selected tags.

CAUTION Failure to define the calibration settings prior to starting the Maintenance module result in the inability to enter the calibration values when entering preventive maintenance details. The **Calibration As Found/As Left** dialog box will not be accessible. Therefore, it is essential that you enter the calibration settings for the selected tags before you schedule a PM job from the browser view.

5. Go back to the Instrument Index Standard Browser view and right-click the selected tags.
6. Associate the tag category and criticality as needed. For details, see Associate Tag Category and Criticality.
7. On the shortcut menu, click **Maintenance > Schedule Preventive Maintenance Activities**.
8. Create the new schedule for the PM activity.
9. Start the Maintenance module and fill out the PM activity details as desired.

At this stage, you can also issue work requests and open repair forms for the instruments as needed.

NOTE When saving PM activity details, SmartPlant Instrumentation prompts you to create a work request if the status of the PM activity is **Failed**. Clicking **Yes** creates the Preventive Maintenance record and opens the **Work Request** dialog box where you can create a repair form.

Schedule Maintenance

1. With the **Maintenance Activities Summary** window open, click  on the **Main** toolbar to start the Instrument Index module.
2. Click **Actions > Browse Index** or click  on the Instrument Index module toolbar.
3. In the **Instrument Index Standard Browser** view, select the instrument for which you want to schedule maintenance. You can select multiple tag numbers if needed.
4. Right-click the selected tags to open a shortcut menu, point to **Maintenance** and select **Schedule Preventive Maintenance Activities**.
5. From the **PM code** list, select the appropriate preventive maintenance code to associate the selected instruments with the required PM code, or click .
6. From the **Assigned workgroup** list, select the work group to which the preventive maintenance activity will be assigned. If the required work group is not available in the list, click .
7. From the **Priority** list, select the required job priority.
8. Set the starting date, frequency, and the PM interval. By default, these values are taken from PM definition.

9. In the **Limit by** section, select either the **Date** or the **Number of PM Activities** option button to choose whether you want to set the number of times this PM activity to be carried out or be limited by a time period.
10. In the **Scheduling Options** section, select how you want to fine-tune the scheduling:
 - **No changes** — the scheduling will remain as you defined it in the **Start date**, **Frequency**, **Interval**, and **Limit by** options.
 - **Schedule jobs for** — fine-tune the scheduling by selecting a specific day of the week, for example every Tuesday of the month.
 - **Schedule jobs for each** — fine-tune the scheduling by selecting a specific day of the month, for example every 15th of the month.
11. Click **Preview** to display the scheduling in the **Preview** data window.

TIP You can also fine-tune the scheduling by editing the PM dates in the **Preview** data window. Click the required PM date and modify the date as needed. All the changes will be implemented after you click **Generate**.
12. Click **Generate** and then click **Close**.
13. Go back to the **Maintenance Activities Summary** window by selecting it from the **Window** menu.
14. In the **Maintenance Activities Summary** window, click  to display the new PM records that you have just scheduled for selected instruments.

Extend Existing Scheduling

1. Start the Maintenance module.
2. In the **Maintenance Activities Summary** window, filter the records as needed.
3. Highlight the required row in the **Preventive Maintenance** data window and do one of the following:
 - Click **Actions > Extend**.
 - Click .
4. On the **Extend Scheduling** dialog box, do the following:
 - a. On the **Take scheduling data from** group box, select one of the following:
 - **Last preventive maintenance activity** — fetch the scheduling data from the last PM activity.
 - **Preventive maintenance code defaults** — fetch the scheduling data from the PM codes.
 - b. On the **Limit scheduling by** group box, select one of the following:
 - **Date** — Sets the scheduling limited by date. Use the spinner to set the required date.
 - **Number of activities** — Sets the scheduling limited by the number of activities. Enter the number of activities in the field provided.
5. Click **Generate**.
6. Click **Close** after the record generation is complete.

Enter Preventive Maintenance Details

1. Start the Maintenance module.
2. In the **Maintenance Activities Summary** window, filter the records that you want to display.
3. In the **Preventive maintenance** pane, select the desired record and do one of the following:
 - Click **Actions > Open**.
 - Click .
4. In the **Work Activity Details** window, from the **Work Status** list, select the required work status if you need change the displayed one.

NOTES

- You can add your own values to the lists in the **Work Activity Details** window using the supporting tables, which you access from the **Tables** menu.
 - The following parameters in this window are not accessible to the repair technician and can be modified only by the maintenance supervisor:
 - **PM code**
 - **Priority**
 - **Created by**
 - **Creation Date**
 - **Last preventive maintenance**
 - **Interval**
 - **Frequency**
5. Assign the required work group by selecting the required option from the **Maintenance staff (scheduled)** list.
 6. Modify the automatically generated work order number if required.
 7. Modify the scheduled date if needed.
 8. Enter the start and completion dates, service, and down time.
 9. From the **Maintenance staff (actual)** list, select the repair technician.
 10. Select the required PM result from the **Results** list.
 11. If the maintenance result is **Failed**, select the failure reason and repair action.
TIP The Failure reason and Repair action lists are available only if the maintenance result is **Failed**.
 12. Type any additional notes you might require in the **Note** data window.
 13. Select the **Calibration Required** check box and click  to open the **Calibration Data Entry** window where you enter the required calibration values. Note that the **Calibration Data Entry** window is accessible only if you have entered appropriate calibration settings for the selected instrument before entering PM activity details.
 14. To change the work status, select the required option from the **Work status** list.
 15. Click  and create a work request if needed.
 16. Click **Yes** when prompted for SmartPlant Instrumentation to create a work request automatically.
TIP SmartPlant Instrumentation prompts you to create a work request when a maintenance activity fails. This happens when you select **Failed** from the **Results** list in the **Work Activity Details** window.

17. Click **Close** in the **Work Activity Details** window to return to the **Maintenance Activities Summary** window.

Performing Breakdown Maintenance

You create new work requests or repair forms from the Instrument Index module. These procedures depend on the level of access rights granted to you. For details, see *Access Rights in the Maintenance Module* (on page 351). Once a new work request or a repair form has been created, it can only be accessed from the Maintenance module **Maintenance Activities Summary** window. The maintenance supervisor can review the work requests and use them to create repair forms. The repair form indicates the work request number that was used to create it. The repair form also includes the problem description. At this stage, it is also possible to indicate that calibration is required. The supervisor then schedules this work for a certain date and assigns the work group that will deal with it. The status of the job at that point is **Scheduled**. After the maintenance work is carried out, the technician responsible for it can fill in the rest of the repair information, and change its status to **Done** or close it. You can customize additional statuses such as **Hold**, **Turnaround**, and so forth to cope with situations and that comply with the conventions at your plant.

NOTE It is recommended to set the required quality system and criticality prior to starting the Maintenance module.

Perform Breakdown Maintenance

The following is the recommended flow of activities for breakdown maintenance:

1. Start the Instrument Index module and open an Instrument Index Standard Browser view.
2. Select the desired instruments (tag numbers) for which you want to enter work requests and start the Calibration module.
3. If the instruments require calibration, enter the required calibration settings for the selected tag numbers. For details, see *Define Tag Calibration Settings* (on page 409).

CAUTION Failure to define the calibration settings prior to starting the Maintenance module results in the inability to enter the calibration values when completing a Repair Form. (The **Calibration As Found/As Left** dialog box will not be accessible.) Therefore, it is essential that you enter the calibration settings for the selected tags before you open Work Requests from the Instrument Index Standard Browser view.

4. Go back to the Instrument Index Standard Browser view and right-click the selected tags.
5. Associate the tag category and criticality as needed. For details, see *Associate Tag Category and Criticality*.
6. In the browser view, right-click the selected tags and click **Maintenance > Repair Form** or **Maintenance > Work Request Form**.
7. Enter the desired repair or work request values.
8. Start the Maintenance module to complete the entry of the required repair or work request values.

Issue a New Work Request

NOTE You can issue new work requests only from the Instrument Index Standard Browser view or from the **Work Activity Details** window if the activity result status was **Failed**.

1. Open an Instrument Index Standard Browser view.
2. Select the required tag number for which you want to issue a work request.

CAUTION Do not select multiple tag numbers in the browser view. This option is available for single instruments only.

3. Right-click the selected tag and from the shortcut menu, click **Maintenance > Create Work Request**.
4. In the **Create Work Request** dialog box, modify the work request number if needed - the software fills in the values automatically.
5. Select the **Typical Instrument** and the **Work Request Reason** from the lists.
6. Type the required work request title.
7. Type the appropriate problem description if needed.
8. Click **Save** and then **Close** to return to the browser view.

TIP You cannot generate a repair form in this case.

Opening a Work Request

You can open work requests from an Instrument Index Standard Browser view or from the **Maintenance Activities Summary** window. However, you can issue new work requests only from the Instrument Index Standard Browser view or from the **Work Activity Details** window if the activity result was **Failed**. It is recommended that you create a new work request from the browser view and then open this new work request record in the **Maintenance Activities Summary** window. Due to the access rights restrictions, some of the work request features are disabled if accessed from the browser view.

If you have been granted full access rights and you are the maintenance supervisor, you can first issue a new work request in the Instrument Index Standard Browser view and then open it in the **Maintenance Activities Summary** window. This enables you to create a repair form. Repair form creation is not available when issuing a new work request, which is only possible from the **Maintenance Activities Summary** window. It is not possible to create a repair form from the work request because all work requests have to be reviewed by the supervisor.

When you issue or open a work request, SmartPlant Instrumentation automatically assigns this record a work request number that you can modify if needed.

Open an Existing Work Request

1. In the **Maintenance Activities Summary** window, filter the records as needed.
2. In the **Work requests** pane, select the required records and click .
TIP All work requests have the **WR** notation preceding the work request number.
3. In the **Work Request** window, modify the work request number and title if needed these are filled in automatically by SmartPlant Instrumentation. If you leave the **Word request title** field blank, the software fills it after you select a typical instrument and a work request reason.
4. Modify or select the typical instrument and the work request reason from the lists. If the required option is not on the list, click **Tables > Typical Instruments** or **Tables > Work Request Reasons** as needed.
5. Type the appropriate problem description if needed.
6. To generate a repair form, click **Actions > Repair Form**.
TIPS
 - This option is not available if you opened the **Work Request** dialog box from an Instrument Index Standard Browser view.
 - This option is accessible only if you have been granted the appropriate access rights. For details, see *Access Rights in the Maintenance Module* (on page 351).
7. Click **Save** and then **Close** to return to the **Maintenance Activities Summary** window.

Create a New Repair Form

1. Open an Instrument Index Standard Browser view.
2. Select the required tag number for which you want to issue a work request.
3. Right-click the selected tag, and from the shortcut menu click **Maintenance > Repair Form**.
NOTE It is also possible to create a new repair form from the **Work Request** window when opening a work request from the **Maintenance Activities Summary** window. You can then open the newly created repair form from the **Maintenance Activities Summary** window. A repair form that you create from a work request retains its **WR** identification number.
4. In the **Repair (scheduled)** group box of the **Repair Form** window, modify the scheduled date and the work order number if necessary.
TIP The **Repair (actual)** options are not available at this stage. The **Repair** options contain the actual work order information.
5. Select a work request reason from the list. If the required option is not available in the list, click the required option on the **Tables** menu.
6. Type the work request title and the problem description.
7. Set the repair priority by selecting the required option from the **Priority** list.
8. From the **Maintenance staff (scheduled)** list, select the appropriate work group from the list or accept the displayed one. If the required option is not available, click the required option on the **Tables** menu.
9. Click **Save** and then **Close** to return to the **Maintenance Activities Summary** window.
In due time, a repair technician can open this repair form from the **Maintenance Activities Summary** window and fill out the information under **Repair (actual)**.

Fill Out a Repair Form

NOTE You can add your own values to the lists in the repair form using the supporting tables, which you access from the **Tables** menu.

1. In the **Maintenance Activities Summary** window, filter the records as needed.
2. In the **Repairs** pane, select the desired records and click .
3. In the **Repair (scheduled)** group box of the **Repair Form** window, do the following:
 - a. Update the work status from the **Work Status** list.
 - b. Modify other values as needed.
4. In the **Repair (actual)** group box, set the start and completion dates.
5. Set the down and repair time.
6. To perform a calibration, do the following:
 - a. Select the **Calibration required** check box.
 - b. Click **Actions > Calibration**.
 - c. Follow the calibration flow of activities. For details, see *Flow of Activities for Entering Tag Calibration Data* (see "Calibration Data Entry Activities" on page 411).
7. From the **Maintenance staff (actual)** list, select the person who is assigned to carry out the repair.
8. From the **Failure reason** list, select the failure reason of the instrument.
9. From the **Outcome** list, select the required item that describes the effect of the current failure, for example partial or complete unit shutdown, and so forth.
10. From the **Repair Action** list, select the value that describes what action has been taken to repair the failure.
11. In the **Comments** data window, type a brief comment if needed.
12. When done, click **Save** and then **Close** to return to the **Maintenance Activities Summary** window.

Maintenance Event Records

For an instrument existing in As-Built or in an engineering company domain it is possible to create records of repairs, tests, inspections, calibrations, and so forth. These records are called maintenance event records. It is possible to associate several maintenance event records with a particular instrument, provided that this instrument belongs to one of the following tag classes: Conventional, Fieldbus, HART, or Profibus. A user with full **Calib. & Maint. Events** access rights can create maintenance event records. After a record is created, it is considered active until a supervisor user with full **Calib. & Maint. Event Supervisor** rights marks the record as complete. Completed records are automatically saved to the database. Only a supervisor can delete completed maintenance event records.

Maintenance event record creation and management is performed in the **Domain Explorer**. In the **Domain Explorer**, all maintenance event records appear under instruments, in the **Maintenance Events** folder. Active records are indicated by . Completed records are indicated by .

When creating a record, the software specifies a name for the record automatically, according to the preferences that you set on the **Preferences** dialog box. For details of the preferences options, see Calibration and Maintenance > General (Preferences). The maximum length of the record name can be sixty characters. Before a record is complete, a user with full **Calib. & Maint. Events** access rights can update the record name by changing the event date.

When creating a maintenance event record, you must select a maintenance even form whose process function is the same as the process function of the source instrument. If you assign a maintenance event form to an instrument type profile in the Instrument Index module, this form is selected automatically when you open the **Maintenance Event Properties** dialog box to create a new record. You need to familiarize yourself with various options of the Specifications module to be able to define and manage forms compatible with maintenance event records. For more details about forms, see Spec Forms. There is one shipped form designed for displaying maintenance event records for level instruments: Displacer Level Switch (form number 93). You need to restore this form first. This form page contains non- editable fields from the COMPONENT table. When regenerating a page for such a form, you can add editable fields from the WORK_ACTIVITY table, and also add non- editable fields from the CALIBRATION_SETTING, COMPONENT, PD_GENERAL, and SPEC_SHEET_DATA tables. If you want to create maintenance event records for instruments whose process function is other than Level, you need to use the **Change Process Function** options of the **Page Editor**. For details, see Select a Different Process Function for a Page.

A page on which you base maintenance event records does not have a title block and, therefore, does not contain any revision data or document number. You cannot use comparison options with maintenance event records. You can create, open and complete one record at a time. It is possible to open and print out several completed records at a time.

Defining and Managing Maintenance Event Records

The following is the recommended flow of activities for defining and managing maintenance event records in SmartPlant Instrumentation.

1. Grant Access Rights for Opening and Completing Maintenance Event Records

The Domain Administrator needs to define access rights for users working with maintenance event records in As-Built or in an engineering company domain. These access rights allow a regular user to create maintenance event records and update the record names. Such a user must have full Calib. & Maint. Events access rights. After a record is created, it is considered active until a supervisor user marks this record as complete. A supervisor user can delete the record if it becomes obsolete. Such a user must have full Calib. & Maint. Event Supervisor access rights. Before granting access rights, the Domain Administrator needs to assign these users to different user groups because access rights in SmartPlant Instrumentation are granted per group. For details about access rights, see the *Administration Module User's Guide, Domain and Project Administration, Access Rights*.

2. Set a Date Format for Maintenance Event Record Names

Use this procedure to set a date format that the software applies to names of maintenance event records when you create such records in the **Domain Explorer**. This procedure can be only be performed by a user that have rights to set or modify preference settings of SmartPlant Instrumentation. For details, see *Set a Date Format for Maintenance Event Record Names* (on page 364).

3. Create a Maintenance Event Record

Use this procedure to create a maintenance event record. This procedure can only be performed by a user with full Calib. & Maint. Event Supervisor access rights. For details, see *Create a Maintenance Event Record* (on page 364).

4. Complete a Maintenance Event Record

Use this procedure to mark a maintenance event record as completed. This procedure can only be performed by a user with full Calib. & Maint. Events access rights. For details, see *Complete a Maintenance Event Record* (on page 365).

Set a Date Format for Maintenance Event Record Names

1. Click **File > Preferences**.
2. In the tree view, click **Calibration and Maintenance > General**.
3. In the **Maintenance event date format** box, enter a valid date format using one of the following examples:

Date Format	Example
mm-dd-yy	01-30-09
mm-dd-yyyy	01-30-2009
dd-mm-yy	30-30-09
dd-mm-yyyy	30-30-2009
d-mmm-yy	30-Jan-09
mmm-yy	Jan-09

NOTES

- SmartPlant Instrumentation only support date formats that are listed in this table.
- You can use one of the following characters as separators: dash (-), slash (/), space, comma, semicolon, colon, or dot.

Create a Maintenance Event Record

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the instrument level in the **Instruments** folder.
2. Click an instrument for which you want to create a maintenance event record. The instrument that you select must belong to one of the following classes: Conventional, Fieldbus, HART, or Profibus.
3. Right-click the **Maintenance Events** folder and on the shortcut menu click **New > Maintenance Event Record**.

NOTE You can create one maintenance event record at a time.

4. Under **Maintenance event date**, define the date of the event.
When you define the date, the software automatically displays the record name in the **Record name** box, according to the preferences you set. For details, see Calibration and Maintenance > General (Preferences).
5. In the **Maintenance event description** box, type and optional description.

The description does not appear automatically on the record sheet. To display the description on the record sheet, you need to regenerate the form page on which the record is based and add the **wa_prob_desc** field in the page.

6. Under **Form number**, select a form whose process function is the same as the process function of the source instrument.
7. Click **OK** to create the record and display it in the **Maintenance Events** folder. It is indicated as an active record by .

Complete a Maintenance Event Record

1. Press F7 to open the **Domain Explorer** and then expand the hierarchy to the instrument level in the **Instruments** folder.
2. Click an instrument containing active maintenance event records, which are indicated by .
3. Click the **Maintenance Events** folder to display the existing records.
4. Right-click a specific record and then, on the shortcut menu, click **Actions > Open Document**.
5. On the **Maintenance Event Record** window menu bar, click **Actions > Mark as Complete**.

NOTES

- In the **Domain Explorer**, completed records are indicated by .
- After a record is marked as completed, you cannot update any settings for the record, only open a record print preview or delete the record.
- To delete obsolete records, right-click one or more records in the **Domain Explorer** and then, on the shortcut menu, click **Delete**. Alternatively, click **Properties** and then delete the selected records one by one from the **Maintenance Event Properties** dialog box. All records in your selection must be either active or completed.

Dimensional Data and Piping

The Dimensional Data for Piping (DDP) module enables you to store and manage three levels of dimensional data: default, working, and vendor. This structure represents the natural flow of information and activities related to dimensional data processing.

You can store and maintain default instrument dimensional data by classifying it per group, manufacturer, or process connection.

The DDP module also enables you to store and maintain dimensional data received from a vendor. This data can also be classified per group, manufacturer, or process connection.

Most significantly, the DDP module provides the means to transfer, store, and maintain certified vendor dimensional data for your instruments. This certified vendor data will then be transferred to your Working data which you use for the actual instruments in your database.

You can use the DDP module to transfer external dimensional data to your SmartPlant Instrumentation database and export of dimensional data from your database to an external 3-D piping design application.

In addition, the DDP module enables you to generate and print out various dimensional data reports that facilitate your dimensional data management. These reports can help you keep track of suspected data, data status, default dimensional data, and vendor dimensional data, and so forth.

NOTE The Dimensional Data for Piping module is available only if it is included in the software license that you purchased from Intergraph.

Principles of the DDP Module

This section provides general guidelines and a recommended flow of activities for the DDP module.

At the initial stage, you should select the appropriate instrument types and define their instrument profiles so that dimensional data handling is clearly specified. Based on the existing dimensional library and practices of dimensional data processing, you also classify the instruments by their type and pre-assigned DDP group by defining this group in the instrument type profile.

Next, define basic instrument parameters in the Instrument Index module or Specifications module, including the instrument manufacturer, model, and process connections.

Group definition coupled with basic information enables the dimensional data designer to provide piping designers with preliminary dimensions filled from the standard dimensional data library based on specified basic instrument parameters.

When requests for quotations are issued and vendor bids are evaluated, more detailed dimensional information supplied by vendors and manufacturers in electronic or hard copy form can be automatically imported or manually entered as vendor data.

This vendor data lets the designer re-evaluate dimensional data, if desired, and issue vendor-certified dimensional data to the piping.

Starting from the moment when the preliminary dimensional data was selected for the instruments, you are able to release this information to the piping design. This can be done in two forms — electronically or by providing a hard copy. Electronic form is an automatic transfer and processing of the structured data to a 3-D CAD system. Alternatively, dimensional data report or dimensional data sheet can be printed out and transferred to the piping design department in paper form.

The status of the dimensional data design can be tracked through the instrument status. You can also view the history information for the selected instrument modifications.

Controlling the modifications of critical definitions, such as a DDP group, instrument type, model, and process connections provides the means to manage and keep track of suspected data during the whole life cycle of the dimensional data design.

Additional Settings

You also have to make some additional settings for the dimensional data. These settings include the following:

- Equipment and equipment type
- Instrument manufacturer
- Instrument model

You access the appropriate dialog boxes from the **Dimensional Data for Piping Module** window and by selecting the desired command on the **Tables** menu. These definitions are actually made in the Instrument Index module and can be accessed from the Dimensional Data for Piping module too.

For a detailed explanation about the above-mentioned settings, see the Instrument Index module documentation.

Start the DDP Module

Use this procedure to start a DDP (Dimensional Data for Piping) module session.

NOTE The DDP module is available only if it is included in the software license that you purchased from Intergraph.

Do one of the following:

- Click .
- Click **Modules > Dimensional Data**.

Preliminary Procedures

There are two preliminary procedures that you have to carry out before you start entering dimensional data. These procedures are:

- Defining the setting for the Dimensional Data for Piping module. To carry out this procedure, you must log on to the Administration module as System Administrator and enter the desired data on the **Dimensional Data Settings Dialog Box**.
- *Associating an Instrument Type with a DDP Group* (see "Associate an Instrument Type with a DDP Group" on page 369)

Define the Process Connection Class/Rating

This procedure enables you to define the desired process connection class / rating. This represents the value of the maximum in-line pressure that the process connection can withstand.

1. In the **Dimensional Data for Piping Module** window, click **Tables > Process Connection Classes**.
2. To create a new process connection class, click **New**.

Type the appropriate process connection class name and description.

TIP You cannot delete a process connection class if it is associated with default, working, or vendor data.

NOTE The connection class name may not be longer than 9 characters so that the DDP Export Utility can include it in the output file.

Define a Process Connection End Preparation

This procedure enables you to define the desired process connection end preparation.

1. In the **Dimensional Data for Piping Module** window, click **Tables > Process Connection End Prep**.
2. To create a new process connection end preparation, click **New**.
3. Type the appropriate process connection end preparation name, design code, and description.

NOTE The end preparation name may not be longer than 9 characters so that the DDP Export Utility can include it in the output file.

Define a Dimensional Group

Defining the dimensional groups is the step in storing and managing dimensional data. You need dimensional groups to store, display, and manage default, working and vendor dimensional data. Moreover, instrument types are also associated with dimensional groups. Hence, new tag numbers whose instrument type is associated with a particular dimensional group will contain dimensional data and this dimensional record will be displayed in the **Working Data** window.

NOTE The Domain Administrator first needs to import DDP library data, including default groups.

1. In the **Dimensional Data for Piping** window, click **Tables > Dimensional Groups**.
2. To create a new dimensional group, in the **Dimensional Groups** dialog box, click **New**.
3. Under **Dimensional Group**, type the group name.
4. Under **Description**, type the group description.
5. Select the **3 Points** and/or **4 Points** check boxes, depending on how many connection points you want to define.
6. In the **CAD Code** field, type the desired CAD code.
7. To define the current group parameters, click **Properties**. See *Defining Dimensional Group Properties* (see "Define Dimensional Group Properties" on page 368) for more details.
8. To assign a picture to the selected group, do the following:
9. Under **Picture**, click **Assign**.
10. In the **Select Group Picture** dialog box, navigate to the desired file, and click **Open**.
11. To assign a dimensional data sheet form to the current group, under **Form**, click **Assign**.

Define Dimensional Group Properties

This procedure enables you to define dimensional data parameters for a given dimensional group. These parameters then appear in the **Default Data** window, the **Working Data** window, and the **Vendor Data** window, as well as in the **Dimensions** section of the **Add Data**, and **Edit Data** dialog boxes.

Thirty parameters are available for each dimensional group. You create an active parameter by typing its description, selecting it for use, and setting its allowed minimum value to be zero or greater than zero.

1. In the **Dimensional Data for Piping Module** window, click **Tables > Dimensional Groups**.
- TIP** At this point, you can modify general properties of the dimensional group. For details, see *Defining Dimensional Groups*.
2. Click **Properties**.
 3. In the **Dimensional Group Properties** dialog box, under **Description**, type the desired parameter description.
 4. To include this parameter in the selected dimensional group, click **Select**.

TIPS

- To select all thirty parameters, select the **Select all** check box.
 - You cannot clear a **Select** check box for a parameter for which the dimensional group is assigned.
5. If you require the selected parameter to have a value greater than zero, select the **Release if Zero** check box.

TIP Clearing **Release if Zero** does not prevent you from placing zeros in this field, but rather prevents the release of dimensional data by disabling the **Ready for Release** option.

6. Click **OK** to accept your definitions and return to the **Dimensional Groups** dialog box.

Duplicate a Dimensional Group

This option enables you to create a new dimensional group based on the same group parameters as the selected group.

1. With the **Dimensional Groups** dialog box open, highlight the desired group.
2. Click **Duplicate** to duplicate the selected group.
3. Click **OK** to accept your definitions, save the group data, and close the dialog box.

Assign Dimensional Data Forms

Assigning a dimensional data form to a dimensional group consists of two stages:

- Assigning a .psr file to a dimensional data form
- Assigning the dimensional data form to a dimensional group

Assign a PSR File to a Dimensional Data Form

1. In the **Dimensional Data for Piping Module** window, on the **Tables** menu, click **Dimensional Data Sheet Forms**.
2. To create a new dimensional data form, do the following:
3. Click **New**.
4. Type the form name and description in the appropriate fields.
5. Select the dimensional data form to which you want to assign a .psr file, and click **Assign**.
6. In the **Select Group Form** dialog box, navigate to the desired .psr file and click **Open**.
7. In the **Dimensional Data Forms** dialog box, click **OK**.

Assign a Dimensional Data Form to a Dimensional Group

1. Click **Tables > Dimensional Groups**.
2. Select the group to which you want to assign a dimensional data form, and under **Form**, click **Assign**.
3. In the **Assign Dimensional Data Form** dialog box, select a form, and click **OK**.

Associate an Instrument Type with a DDP Group

This option enables you to set the default dimensional group for new tag numbers that are associated with dimensional data. Note that this procedure is performed in the Instrument Index module.

1. In the **Instrument Index Module** window, click the **Tables > Instrument Types**.
2. From the **Process function** list, select the process function.
3. In the data window, select the instrument type.
4. Click **Profile**.
5. On the **General** tab, under **Dimensional data**, do the following:
6. Select the **Include dimensional data** check box to associate the selected instrument type with dimensional data.

7. From the **Group Name** list, select the dimensional group with which dimensional data will be associated by default.
8. Click **OK** to close the **Instrument Type Profile** dialog box and then click **OK** to close the **Instrument Types** dialog box.

Vendor Data

Vendor data is used for data validation and verification purposes of dimensional data for piping design. You use Vendor data to certify the Working data prior to its release to piping.

Instead of inserting raw Vendor data manually, use the Import Utility to import a large amount of raw vendor information. After importing the Vendor data, you revise the raw vendor data to assign it to a dimensional group, verify the instrument tags, and, if desired, manually modify the vendor data details.

Sometimes multiple records for the same instrument can exist. This can happen if you import vendor data from several manufacturers. You can copy vendor data to the Working data if the manufacturer, model, the Dimensional Group, and the process connection values match the Working data values.

You can also indicate whether the imported data has been copied to the Working data. You can reset this indication every time import is performed for a specific record, thus creating a clear indication if the latest information has been copied to the Working data.

Managing Vendor data involves editing it and entering new dimensional vendor data for a selected instrument.

Vendor data is displayed in the **Vendor Data** window.

Display and Access Vendor Data

1. With the **Dimensional Data for Piping Module** window open, do one of the following;
 - Click .
 - Click **View > Vendor Data**.
2. In the **Select Dimensional Group** dialog box that opens, select the desired dimensional group and click **OK**.

The **Vendor Data** window opens displaying the existing vendor data records.

You can customize the display and layout of the data in the **Working Data** window. The following options are available:

- Determining the fields to be displayed
- Sorting the data
- Filtering the data

Enter Vendor Data

This feature allows you to enter vendor data manually. It is useful when adding a few records. If you want to enter numerous records, use the Import Utility.

1. Open the **Vendor Data** window and do one of the following:
 - Right-click anywhere in the **Vendor Data** window and click **New Vendor Data** on the shortcut menu.
 - Click **Actions > New Vendor Data**.
2. Beside **Tag number**, click **Browse** and then do the following:
3. In the **Enter Tag** dialog box, type in the desired tag number and click **OK** or click **Find** to find the desired tag number.
4. In the **Find Tag** dialog box, enter the desired search parameters and click **Find** to find the desired tag number.
5. Under **Search Results**, highlight the desired tag number and click **OK** to return to the **New Vendor Data** dialog box where this tag number is displayed in the **Tag name** field.
6. Select the desired instrument manufacturer from the **Manufacturer** list.
7. Select the desired instrument model from the **Model** list.
8. Type the desired full and dry weight values and select the weight unit of measure from the **Weight UOM** list.
9. In the **Revision** field, type the revision number as needed.
10. Enter the default INLET and OUTLET values: type in the **Size** values and select the appropriate values from the **Class** and **End preparation** lists.
11. Enter the default #3 and #4 process connection values if needed: type in the **Size** values and select the appropriate values from the **Class** and **End preparation** lists.
12. In the **Dimensions** section, type in the default dimensional properties. Note that the **Dimensions** field names appear as you have defined them in the **Dimensional Groups** dialog box.
13. Type the **Full** and **Dry Weight** values.
14. Select the weight unit of measure from the **Weight UOM** list.
15. Click **OK** to save the data and close the dialog box.

Working Data

The **Working Data** window displays existing Working dimensional data for tag numbers entered in the Instrument Index module. Once you create a new tag number whose instrument type is associated with dimensional data and a dimensional group, the dimensional data for this tag will be displayed in the **Working Data** window.

You can also add more records to the Working data by copying from Vendor data or by copying preliminary default data from the Default Library if the Dimensional Group, manufacturer, model, and the process connection values match the Working data.

Managing Working data involves editing it, modifying the data status, entering the piping design area, and finally generating and printing out a Dimensional Data Sheet for a particular instrument.

You access Working data in the **Working Data** window.

Display and Access Working Data

This feature enables you to store and manage Working dimensional data.

1. With the **Dimensional Data for Piping Module** window open, do one of the following;
 - Click .
 - Click **View > Working Data**.
2. In the **Select Dimensional Group** dialog box, highlight the desired dimensional group and click **OK**.

The **Working Data** window opens displaying the existing working data records for the instruments whose instrument type is associated with dimensional data and the current dimensional group.

TIP You can customize the display and layout of the data in the **Working Data** window. The following options are available:

- Determining the fields to be displayed
- Sorting the data
- Filtering the data

Edit Working Data

You can edit the working data either in the **Working Data** window or in the **Working Data Properties** dialog box. Note however, that not all the fields are accessible for editing in the **Working Data** window. Therefore, you should edit the data in the **Working Data Properties** dialog box.

1. With the **Working Data** window open, highlight the desired tags and do one of the following to open the **Working Data Properties** dialog box:
 - Right-click the selected tag numbers and click **Working Data Properties** on the shortcut menu.
 - Click **Actions > Working Data Properties**.
2. Enter the default **INLET** and **OUTLET** values and from the **Units of Measure** list choose the required unit of measure.
3. Type in the **Size** values and select the appropriate values from the **Class** and **End preparation** lists.
4. Enter the default **#3** and **#4** process connection values and from the **Units of Measure** list choose the required unit of measure.
5. Type in the **Size** values and select the appropriate values from the **Class** and **End preparation** lists.
6. Under **Dimension values**, type the default dimensional properties. Note that the **Dimensions** field names appear as you have defined them in the **Dimensional Groups** dialog box.
7. Type the **Full** and **Dry Weight** values.
8. Select the weight unit of measure from the **Weight UOM** list.
9. Click **OK** to save the data and close the dialog box.

Piping Design Area

This option enables the piping designers to enter 3-D CAD design model or piping design area for the selected instrument. You can select multiple records from the Working **Data window** and enter or modify the piping design area as needed.

1. In the **Working Data** window, highlight the desired instruments and do one of the following:
 - Right-click the selected instruments and click **Piping Design** on the shortcut menu.
 - Click **Actions > Piping Design**.
2. Type in the desired design area name in the **Design Area** field and click **Save**.
3. Select the **Copy to all tags** check box to copy the design area name you have just entered to all the selected instruments or click **Preview** or **Next** to open the previous or the next record.
4. Click **OK** to accept the values and close the dialog box.

NOTE All the other fields in this dialog box are read-only.

Copy Working Data to the Default Library

This option enables you to copy working dimensional data to the Default Library. The copied record includes all its definitions: the dimensional group, manufacturer, model, and the process connection values.

You can select only a single instrument at a time. This instrument will serve as a default example. For each selected instrument, the software can then search a default record with corresponding dimensional group, manufacturer, model, and process connection values.

1. In the **Working Data** window, highlight the instruments whose working data you want to copy to the Default Library and then do one of the following:
 - Right-click the selected tag numbers and click **Copy Working to Default** on the shortcut menu.
 - Click **Actions > Copy Working to Default**.
2. Examine the information in this **view-only** dialog box and click **Copy** to copy the working data to the Default Library.

TIP The **Copy** button is disabled if no matching values are found.

Copy Default Data to Working Data

This feature allows you to copy records with preliminary default dimensional data into the working data of the selected instrument. Copying default data from the Default Library is only possible if the dimensional group, manufacturer, model, and the process connection data match the working data.

You can select numerous instruments whose default data you want to copy. For every selected tag number, the software can then search for values matching the working data. Only after matching dimensional group, manufacturer, model, and process connection values are found, the software copies the default data.

After successfully copying default data, the status of the copied records is set as **Preliminary** and its revision number as **0** (zero). The revision number will then be incremented from the last preliminary status by one, for example, **Preliminary 1**.

1. In the **Working Data** window, highlight the tag numbers whose default data you want to copy and then do one of the following:
 - Right-click the selected tag numbers and click **Copy Default to Working** on the shortcut menu.

- Click **Actions > Copy Default to Working**.

The software searches for instruments with matching dimensional group, manufacturer, model, and process connection values. Once matching values are found, the **Copy Dimensional data from Default Data** dialog box opens:

2. Examine the displayed data:

- **Instruments selected** — this field shows how many tag numbers you selected whose default dimensions you want to copy.
- **Instruments found** — this field shows how many tag numbers in the working data match the values in the default data.
- **Instruments with incomplete data** — this field shows how many tag numbers in the working data have incomplete data.

A progress bar opens under the data status indication showing you the data copying progress. For every instrument with matching values, default dimensional values are copied.

3. Click **Copy** to start copying the data and monitor the process in the progress bar under the data status indication.

NOTES

- If the instrument contains no working data, the copied default dimensions will be marked as **ready for release**.
- If the instrument contains working data, you will be prompted if you want the working data to be overwritten by the default data. The copied dimensions will be marked as **suspected**.

Copy from Vendor Data to Working Data

This feature allows you to copy certified vendor data to the working data of a selected instrument. Copying vendor data is only possible if the dimensional group, manufacturer, model, and the process connection data match the working data. You can select numerous instruments for which you want to copy vendor data. For each selected instrument, the software can then search for matching dimensional group, manufacturer, model, and process connection values.

1. In the **Working Data** window, highlight the tag numbers for which you want to copy vendor data and then do one of the following:
 - Right-click the selected tag numbers and click **Copy from Vendor to Working** on the shortcut menu.
 - Click **Actions > Copy from Vendor to Working**.

TIP The software searches for instruments with matching dimensional group, manufacturer, model, and process connection values. Once matching values are found, the **Copy from Vendor to Working** dialog box opens.

2. On the **Copy Vendor to Working** dialog box, examine the data displayed in the following boxes:
 - **Instruments selected** — This box shows how many tag numbers you selected for which you want to copy vendor dimensions.
 - **Instruments found** — This box shows how many tag numbers in the working data match the values in the vendor data.
 - **Instruments with incomplete data** — This box field shows how many tag numbers how many tag numbers have incomplete vendor data, incomplete working data, or data mismatch in the fields that must be identical in both vendor and working data.

IMPORTANT To be able to copy vendor data to working data, make sure that the following vendor data properties match the corresponding working data properties: inlet, outlet, #3, and #4 connection points and their units of measure, instrument manufacturer and model.

3. In the **Set Status** section, select the dimensional data status of the working data as desired: **Preliminary**, **Design**, or **Certified**.
4. Click **Copy**.

NOTES

- For every instrument with matching values, vendor dimensional values are copied including the vendor revision number. This data is marked as **copied to working data**.
- If the instrument contains no working data and all desired dimensional data is defined, the data is marked as **ready for release**.
- If the instrument contains working data, you are prompted if you want the working data to be overwritten by the vendor data. The copied dimensions are marked as **suspected data** and **copied from vendor**.

Modify Working Data Status

This option enables you to display and modify the current status of Working dimensional data for the selected instrument. Three status stages are available: Preliminary, Design, and Certified. Each change of status results in the revision number being set to 0. Within each status, the revision number (from zero to 99) indicates the progress of the dimensional data handling. An instrument status can be described as a combination of these two parameters: the status of the Working dimensional data and the revision number, for example PRELIMINARY 1 or DESIGN 5.

NOTES

- When you modify the data status because of any dimensional data management, you will be able to reset the **suspected data** flag and mark the data as correct and ready for release to piping.
 - You can select multiple instruments for status modification.
1. In the **Working Data** window, highlight the instruments whose working status you want to modify and then do one of the following:
 - Right-click the selected instruments and click **Modify Status and Revision** on the shortcut menu.
 - Click **Actions > Modify Status and Revision**.
 2. On the **Status** tab, click **Set status** and select the desired status from the **Status** list. The list displays the selected status and becomes unavailable until you click **Set status** again.
 3. Click **Raise to next revision** to raise the revision number if needed.
 4. Click in the **Description** text box and type a short status description.
 5. Select the **Reset Suspect Flag** check box to remove the suspected data status for the current instrument.
 6. Click **Save** to save your settings.
 7. Click **Next** or **Previous** to display the next or previous selected tag number.
 8. Click **OK** to close this dialog box or click the **History** tab to open the **History** tab folder.

Regenerate Dimensional Data Sheets

This procedure can be run on a single tag or multiple tags at the same time. You must carry out this procedure first to be able to use the **SmartPlant** menu **Revise** command when working with SmartPlant Foundation.

1. In the Dimensional Data for Piping module open the **Working Data** window.
2. Select the required tag.
TIP Use the Shift or Ctrl keys and click on the required tags to select more than one tag.
3. Right-click, and from the shortcut menu select **Regenerate Dimensional Data Sheets**.
4. At the prompt, click **OK**.

Manage Default Dimensional Data

This feature enables you to store and manage preliminary dimensional data for your instrumentation design. You can use standard (default) dimensions of known manufacturers who are likely to supply the instruments. This Default data is stored in the Default Library which allows you browse through the data, modify it, and copy it to the Working data as needed. Once sufficient Default data is available, the DDP designer can provide this data to the piping designers for their preliminary design. The Default data will then be replaced or modified according to the actual dimensional data received from the manufacturers.

CAUTION Note that you must first define the appropriate dimensional group, the process connection data, and the manufacturer before you can start entering default data. Remember that only one set of default dimensional data can exist for a domain.

Enter Default Data into the Default Library

Use this procedure to add default dimensional data for a given dimensional group.

1. With the **Dimensional Data for Piping Module** window open, do one of the following;
 - Click **View > Default Library**.
 - Click .
2. In the **Select Dimensional Group** dialog box, highlight the desired group and click **OK**.
CAUTION If you select **All Groups**, you will not be able to edit the data and no process connection values will be displayed.
3. Click **Actions > New Default Data**.
4. In the **Default Data Properties** dialog box, select the desired instrument manufacturer from the **Manufacturer** list.
5. Select the desired instrument model from the **Model** list.
6. Enter the default **INLET** and **OUTLET** values: type in the **Size** values and select the appropriate values from the **Class** and **End preparation** lists.
7. Enter the default **#3** and **#4** process connection values if needed: type the **Size** values and select the appropriate values from the **Class** and **End preparation** lists.
8. In the **Dimensions** section, type in the default dimensional properties. Note that the **Dimensions** field names appear as you have defined them in the **Dimensional Groups** dialog box.
9. Type the **Full** and **Dry Weight** values.
10. Select the weight unit of measure from the **Weight UOM** list.
11. Click **OK** to save the data and close the dialog box.

Edit Default Dimensional Data

Use this procedure to edit default data for a given dimensional group.

1. Open the **Default Data** window for the desired dimensional group. For details, see *Entering Data into the Default Library* (see "Enter Default Data into the Default Library" on page 376).
2. Right-click the entries that you want to edit, and on the shortcut menu, click **Default Data Properties**.
3. Select the desired instrument manufacturer from the **Manufacturer** list.
4. Select the desired instrument model from the **Model** list.
5. Enter the default INLET and OUTLET values and from the **Units of Measure** list choose the required unit of measure.
6. Type in the **Size** values and select the appropriate values from the **Class** and **End preparation** lists.
7. Enter the default **#3** and **#4** process connection values and from the **Units of Measure** list choose the required unit of measure.
8. Type in the **Size** values and select the appropriate values from the **Class** and **End preparation** lists.
9. In the **Dimensions** section, type in the default dimensional properties. Note that the **Dimensions** field names appear as you have defined them in the **Dimensional Groups** dialog box.
10. Type the **Full** and **Dry weight** values.
11. Select the weight unit of measure from the **Weight UOM** list.
12. Click **OK** to save the data and close the dialog box.

Customize the Default Library Display and Layout

You can customize the display and layout of the data in the Default Library. The following options are available:

- Sorting the data
- Filtering the data
- Determining the fields to be displayed

For a detailed explanation on how to carry out these procedures, refer to the Browser module, Define a View Profile.

Services and Reports

This section contains topics about generating dimensional sheet reports.

Generate a Dimensional Data Sheet

This feature enables you to produce a dimensional data sheet for a selected instrument. The dimensional data sheet can complement or serve as an alternative for the automatic 3-D CAD piping design.

A dimensional data sheet includes general tag information, dimensional data of the specific group, and the graphic schematic or detailed presentation of the instrument and its dimensions.

1. In the **Working Data** window, highlight the desired instrument and do one of the following:
 - Right-click the selected instrument and click **Dimensional Data Sheet** on the shortcut menu.
 - Click **Actions > Dimensional Data Sheet**.
2. In the **Dimensional Data Sheet** window that opens, you can do the following:
 - Enter or modify the values in the dimensional data sheet fields. Click in the appropriate field to type a value or select a value from a list. Note that not all the fields are accessible for editing.
 - Manage and enter dimensional data sheet revisions. Click  to open the **Revisions** dialog box where you can enter and modify revisions.
 - Save the dimensional data sheet as an external file, click .
 - Print out the dimensional data sheet, click .

TIP You can also preview and print dimensional data sheets in batch mode. For details, see *Printing Dimensional Data Sheets* (on page 380).

Track the Data Status History

This option allows you to track the working data status history.

1. With the **Modify Status** dialog box open, click the **History** tab to open the **History** tab folder.
2. Examine the information and click **Edit** to type in any appropriate description if necessary.
3. Click **OK** to accept the values and close the dialog box.

Generate DDP Reports

You can generate and print out the following reports.

Report	Description
Group List	Contains a detailed list of all existing Dimensional Groups and their parameters, such as group name, description, associated picture file, cad group identifier, and dimensional parameter definitions.
Dimensions List	Displays a detailed list of all dimensional data for each instrument that is associated with dimensional data.

Report	Description
Suspected Data List	Presents a list of all the instruments for which dimensional data must be re-evaluated due to some changes made to the instrument definitions. For example, if the process connection or any other key parameter has been changed, the data is marked as suspected. This report shows the cause of the suspected data flag.
Status History	Contains all the dimensional data status history per all Tags whose dimensional data status has been changed.
Suspected Data History	Displays a list of all Tags whose dimensional data was or still is marked as suspected. The report also presents the reasons for the suspected data indication.
Default Library List	Contains a list of general default definitions for the elements of the Default Library.
Default Dimensional Data	Lists detailed definitions of the elements in the Default Library.
Vendor Dimensional Data	Contains a detailed list of vendor dimensional data.
Data Sheets	Allows you to display data sheets associated with specific tag numbers.
Empty Sheet Forms	Allows you to display field names and dimensional properties only, without displaying the values. An empty sheet form also displays a dimensional group diagram.

Generate a Group List, Dimensions List, Default Library List, Default Dimensional Data, Or Vendor Dimensional Data Report

- With the Dimensional Data for Piping Module window open, on the Reports menu, click a report menu command.

Generate A Suspected Data List, Status History, Or Suspected Data History Report

1. With the **Dimensional Data for Piping Module** window open, on the **Reports** menu, click the appropriate menu command.
2. In the **Report Parameters** dialog box, in the **Report Per** section, select if you want the report to include data for all the instruments associated with dimensional data or just for the selected groups.
3. In the **Show data for** section, select if you want the report to be generated for the current plant or the current unit, and click **OK**.

Printing Dimensional Data Sheets

Use this procedure to open a print preview of one more dimensional data sheets and print dimensional data sheets in batch mode.

Batch preview and printing is available either from the **Dimensional Data for Piping Module** window or from the **Working Data** window.

Print Dimensional Data Sheets From the Dimensional Data for Piping Module Window

1. Click **Reports > Data Sheets**.
2. In the **Select Tags for Printing Dimensional Data Sheets** dialog box, find the desired instruments.
3. Under **Search results**, select source tag numbers.
4. Click **OK**.

Print Dimensional Data Sheets From the Working Data Window

1. In the **Working Data** window, select one or more instruments.
2. Do one of the following:
 - Click **Actions > Print Dimensional Data Sheets**.
 - Right-click, and from the shortcut menu click **Print Dimensional Data Sheets**.
 - Click .

TIP Clicking  only prints the instrument list displayed in the **Working Data** window.

Print Empty Sheet Forms

Use this procedure open a print preview and print one more empty sheet forms. An empty sheet form is a form that does not contain any values. Such a form only displays the names of the fields and dimensional properties, and also display the dimensional group diagram. Each dimensional group is associated by default with one sheet form.

1. Click **Reports > Empty Sheet Forms**
2. In the **Print Empty Sheet Forms** dialog box, select one or more dimensional groups.
3. Click **OK**.

SECTION 37

Construction Module

The Construction module is used to define contracts for installation activities that are performed by external contractors. This involves controlling and monitoring the contracts and installation tasks assigned to various external contractors, in order to enable plant start-up to take place on time. An installation task can be any activity or group of activities that would be applied to an instrument to make it operable.

Each item (instrument tag, wiring scheme, and so forth.) requires a specific set of activities. The module matches the various activities to each contractor by listing them according to categories (fields of operation). Each activity is defined as an **index**, which you can assign to specific items when required.

You can track the status and progress of activities performed by creating index revisions, viewing and printing reports of each revision for a particular index.

For example, the following activities could be involved in the installation of electronic 2-wire transmitters:

Mechanical implementation:

- Instrument stand installation
- Physical instrument installation
- Impulse piping connection

Electrical implementation:

- Wiring of instruments to junction boxes
- Wiring of instruments to marshaling racks
- Wiring of instruments to control system

Instrument implementation:

- Calibration of instruments
- Instrument wiring continuity check
- Configuration of control system
- Check of instrument operation
- Loop checks
- Control system operation check

Only on completion of all these activities can the instruments be fully commissioned and operated.

See also:

Activate the Construction Module (see "Activate the Construction Module" on page 384)

Activate the Construction Module

To work with the Construction Module you must first activate the module from the SmartPlant Instrumentation Preferences.

To activate the module,

1. Select **File > Preferences**.
2. On the **Preferences** dialog box, click  (Module Icons).
3. From the **Available** icons column do one of the following:
 - Select  and click the **Add** button.
 - Double-click .
4. The  appears in the Module toolbar.

Construction Flow of Activities

A typical flow of activities for installation consists of the following stages (the procedures for each stage are described in more detail in the relevant sections):

1. Populating the following supporting tables:
 - **Contractors**
 - **Contracts:** One or more contracts can be defined per contractor.
 - **Installation Index Categories:** SmartPlant Instrumentation provides three default installation index categories: Instrument, Mechanical, and Electrical. You can edit these if required, and you can also add up to two more categories of your own.
 - **Installation Indexes:** One or more installation indexes can be associated with each contract. Each installation index must belong to an installation category. Within each category, there is no limit to the number of installation indexes you can define.
2. Assigning the installation indexes to the desired items (instrument tags, panels, or cables) in the appropriate Construction Browser View.
3. Issuing reports. For each installation index, the following classes of reports can be viewed and printed:
 - **Working report:** Working (current) reports can be accessed from the **Installation Index Manager** or by selecting filtered data to display in the Browser and previewing the reports there.
 - **Formal report:** Reports related to formal issues can be accessed from the **Installation Index Manager Archive**.
 - In both classes, a complete report of the items assigned to the installation index can be displayed, or where relevant, you can issue a change summary report showing the differences between the current situation or selected revision and the previous revision.
4. Retrieving archived reports.

Definitions in the Construction Module

In order to implement the installation, the activities must be defined. The following elements of the Construction module are involved in specifying the activities:

- **Installation Contractors:** The companies responsible for carrying out the different installation tasks.
- **Installation Contracts:** The documents used to specify the details and conditions for each task or group of tasks. One or more contracts can be defined for each contractor.
- **Installation Categories:** Each category can represent a discipline of operation (for example, three default categories of Installation Indexes are defined: Instrument, Electrical, and Mechanical).
- **Installation Indexes:** An installation index is an index number within an installation category associated with a specific installation task. Each task may consist of one or more activities.
- **Items:** A SmartPlant Instrumentation item can be an instrument tag, a panel, or a cable to which an installation index can be assigned. Within each category, only one index can be assigned to a particular item.
- **Formal Issue (Revision):** A formal issue is a revision that is assigned to an installation report when the contract data is stable and is ready to be sent to the client for approval. The revision can then be used as a benchmark for monitoring any changes that may be made later.
- **Archive:** After creating a formal issue for an installation index, the reports containing the details of that index are stored in the Installation Index Manager Archive. From there you can retrieve existing installation reports showing archived information, or reports of the changes that were made to the installation index since the previous revision.

Installation Index Category Definitions

All new installation indexes that are defined must belong to an installation index category. There are three default categories (editable) that are provided as standard: Instrument, Electrical, and Mechanical, and you have the option to add two further installation index categories if required.

Edit an Installation Index Category

NOTE You may only edit installation index categories that do not have any indexes assigned to them.

1. Click **Tables > Category**.
2. In the **Index Categories** dialog box, select the desired row, and then click **Edit** to open that row for editing.
3. Edit the code in the **Prefix** field (up to 5 characters) and **Name** fields if required (you can enter a name of up to 20 characters in the **Name** field).
4. Do one of the following:
 - To make the new installation index category available in the **Installation Index Manager** window and in the header columns for the appropriate Browsers, select at least one of the check boxes:
 - **Wiring Category:** This makes Cable, Panel, and Wire reports available for all items assigned to installation indexes in this category.
 - **Instrument Category:** This makes an Instrument report available for all items assigned to installation indexes in this category.

- To exclude the new installation index category from the lists in the **Installation Index Manager** window and in the header columns for the appropriate Browsers, clear all the check boxes.
5. Click **OK**.

Edit an Installation Index

There may be occasions where you will want to update or change an installation index definition. Editing an installation index may involve changes to the name, description, or contract for the index. After editing an index name, the new name will appear in the appropriate index field in the Browser Views for instrument tags where the original index was assigned.

1. Click **Tables > Installation Index**.
2. In the **Installation Indexes** dialog box, in the data window, select the required installation index, and then click **Properties**.
3. Edit the data in the appropriate fields (**Number**, **Description**, and **Contract Number**) as required.
TIP Installation Index numbers must be unique.
4. Repeat steps 2 through 4 for each installation index you want to edit.
5. Click **OK** to save your changes and close the dialog box.

Delete an Installation Index

When an installation index is no longer needed, you can delete it, provided it has not already been assigned to an item in one of the Browser Views.

1. Click **Tables > Installation Index**.
2. In the **Installation Indexes** dialog box, in the data window, select the index that you want to delete.
3. Click **Delete**.
The installation index is deleted.
4. Repeat steps 2 and 3 for each installation index you want to edit.
5. Click **OK** to save your changes and close the dialog box.

View and Modify Assigned Installation Indexes

After assigning an installation index to an instrument tag, you can open a filtered browser view based on the currently selected index in the **Installation Index Manager** window. The filtered list enables you to view and modify with ease those items (instrument tags, panels, cables, or wires) that already have a particular index assigned to them.

NOTE If you change the electrical index in one of the report views — tag, panel, or cable — SmartPlant Instrumentation updates the electrical index of all the relevant items.

1. In the **Installation Index Manager** window, select the required installation index category from the list.
TIP The list of installation indexes for the selected category is displayed in the lower data grid.
2. In the **Report for** field, select the required report (if more than one report is available for the selected category).

TIP Where several reports are available for a particular installation index, the revision data displayed in the lower data window will depend on the report you select. Thus for the same installation index, different reports may show different revisions.

3. In the lower data window, select the required installation index from the list.
4. Do one of the following:

- Click .
- Click **Actions > Open Filtered Browser**.

The appropriate **Browser View** window opens for the report you selected, for example, the Panel Terminations Report. All panels in the database that are linked to the selected installation index are displayed.

5. In an **Index** column, modify the index selection as required.
6. Do one of the following:
 - Close the window.
 - Click .
 - Click **Actions > Refresh**.
7. At the prompt, click **OK** to save the changes.

Construction Options

The procedures described in this section explain how to create, maintain, and use installation indexes for tracking the tasks performed by contractors.

Construction - Example Scenario

Using the installation of electronic 2-wire transmitters as a basis, the following example shows the kind of data you would enter:

1. In the **Contractors** dialog box, define contractors as follows:
2. Contractor for mechanical tasks.
3. Contractor for electrical tasks.
4. Contractor for instrument- related tasks.
5. Define one contract number for each contractor.
6. Define installation indexes in each installation category (using the default categories) as follows:
7. **Instrument category**
 - **I1:** Instrument calibration only.
 - **I2:** Instrument calibration and various checks.
8. **Electrical category**
 - **E1:** Wire to JB only.
 - **E2:** Wire to JB and MR.
 - **E3:** Wire to MR only.
 - **E4:** Wire to MR and CS.
 - **E5:** Wire to CS only.
9. **Mechanical category**
 - **M1:** Physical installation and connections.
 - **M2:** Connect impulse piping.

10. Assign the installation indexes to the required instrument tags (in the appropriate Construction Browser View), for example, the Instrument Installation Index View is displayed, and indexes I1, E2, and M1 are assigned to a number of flow transmitters.
11. Generate a formal issue for each installation index.
12. Change installation indexes that were wrongly assigned to one of the flow transmitters as follows:
 - I1 to I2.
 - E2 to E1.
13. Create working and formal reports for installation indexes I1, I2, E1, and E2 to show the current statuses of the installation indexes and their statuses prior to the above changes.
14. Generate the following new formal issues:
 - Revision for installation indexes I1 and I2.
 - Revision for installation indexes E1 and E2 (report for wires).

TIP Revisions are performed per installation index and per report, and so in this example, you would create a revision once all the changes for the particular installation index have been made for the appropriate instrument tags.
15. Change the electrical installation index for another flow transmitter from E2 to E3 and generate formal issues for both of these installation indexes (report for wires).
16. Preview and print formal change summary reports from the revision archive for the last formal issue on installation indexes E2 and E3.

View Installation Indexes in the Installation Indexes Dialog Box

There are two locations from which you can view the installation indexes:

- In the **Installation Index Manager** window, you can view a list of the indexes in each category by selecting the category from the **Index category** list.
 - In the **Installation Indexes** dialog box, you can view a list of the indexes in each category, or from all the categories.
1. Click **Tables > Installation Index**.
 2. In the **Installation Indexes** dialog box, do one of the following:
 - To view the installation indexes in a particular category, from the **Index category** list, select the required category.
 - To view the installation indexes in all categories, select **Show all index categories**.
 3. When done, click **Cancel** to close the dialog box.

Add an Installation Index

Adding an installation index requires the following actions:

- Create the contractors in the supporting table.
- Create the contracts in the supporting table.
- Ensure that at least one installation index category is active.
- Create the installation index itself.

Add an Installation Index

1. Define your contractors as follows:
2. Click **Tables > Contractor**.
3. In the **Contractors** dialog box, click **New**.
4. In the new line, enter a name (up to 20 characters) and a definition (up to 40 characters) in the appropriate fields.
5. Repeat steps b) and c) for each contractor you want to add.
6. Click **OK** to save your changes and close the **Contractors** dialog box.
7. Define your contracts as follows:
8. Click **Tables > Contracts**.
9. In the **Contracts** dialog box, click **New**.
A new line appears.
10. In the **Contractor Name** field, select a contractor from the list.
11. In the **Contract Number** field, type a number for the contract.
12. Repeat steps b) through d) for each contract you want to add.
13. Click **OK** to save your changes and close the dialog box.
14. Click **Tables > Category**.
15. In the **Index Categories** dialog box, check that the category definitions meet your requirements, otherwise do one of the following:
 - Create a new installation index category (for details, see *Adding an Installation Index Category* (see "Add an Installation Index Category" on page 390)).
 - Edit an existing installation index category (for details, see *Editing an Installation Index Category* (see "Edit an Installation Index Category" on page 385)).
16. Create the installation index as follows:
17. Click **Tables > Installation Index**.
18. In the **Installation Indexes** dialog box, from the **Index category** list, select a category.

TIP Selection of an installation index category is a prerequisite for defining an installation index. If no category is available in the list, you can create one by clicking  to open the **Index Categories** dialog box. Select at least one of the check boxes: **Wiring Category** or **Instrument Category** for one or more of the installation index categories.
19. Click **New**.

TIP The category prefix is entered by default as the prefix for the Installation Index number.
20. Type the installation index number and description in the appropriate fields.

TIP Installation Index numbers must be unique. If you do not modify the default, you will be prompted to keep it on saving.
21. From the **Contract number** list, select a contract.

22. Repeat steps c) through e) for each installation index you want to add.
23. Click **OK** to save your changes and close the dialog box.

Add an Installation Index Category

There are two user-definable installation index categories. This section explains how to create and use them.

1. Click **Tables > Category**.
 2. In the **Index Categories** dialog box, select an empty row and click **Edit** to open that row for editing.
 3. Enter a short code in the **Prefix** field (up to 5 characters) and type a name (up to 20 characters) in the **Name** field.
 4. In order to make the new installation index category available in the **Installation Index Manager** window and in the header columns for the appropriate Browsers, select at least one of the check boxes:
 - **Wiring Category**: This makes Cable, Panel, and Wire reports available for all items assigned to installation indexes in this category.
 - **Instrument Category**: This makes an Instrument report available for all items assigned to installation indexes in this category.
- TIP** Selection of both check boxes makes all four reports available.
5. Click **OK**.

Assign Installation Indexes to Items

You will remember that each installation index represents a particular activity performed by a particular contractor. For every item (instrument tag, panel, cable, or wire), you can assign one installation index in each of the active categories. Since a contractor may perform the same activity on more than one item, you can assign the same installation index in a given category to several items of the same type, such as instrument tags. For example, you may want to assign the same electrical installation index to all the instrument tags whose process function is measuring pressure.

At this stage, it is assumed that you have defined all your installation indexes as described in the previous sections.

1. With the **Installation Index Manager** window open, select the required installation index category from the **Index category** list.

The list of installation indexes for the selected category is displayed in the data window.
 2. Beside **Report for**, select the required report (if more than one report is available for the selected category).
 3. Do one of the following:
 - Click .
 - Click **Actions > Open General Browser**.
- TIP** The appropriate **Browser View** window opens for the report you selected. For example, if you selected the Electrical Index, and you chose Panels in the **Report for** field, the Panel Terminations Report will be displayed. In the default category definition, the Panels report (as part of the Wiring Category) only appears in the Electrical Index, and therefore only the **Electrical Index** column will appear in the Browser View. In this view, all the panels in the database are displayed.
4. In the **Electrical Index** column, select the required index from the list for each instrument tag as required.

5. Do one of the following:
 - Close the window.
 - Click .
 - Click **Actions > Refresh**.
6. At the prompt, click **OK** to save the changes.

SECTION 38

Reports, Revisions and Archives

This section contains topics about generating reports, revisions, and archives.

In This Section

Generate Reports For An Installation Index Revision Reports.....	393
Construction Module Revisions	393
Maintaining a Revision Archive	394
View an Archived Installation Index.....	394
Edit an Archived Revision.....	395

Generate Reports For An Installation Index Revision Reports

You can display and print a report for each installation index revision you create. Two types of report are available: a general report showing details of all the items in the selected revision, and a change report showing only the differences between the current revision and the previous one.

1. Follow steps 1-3 in *Viewing and Editing Archives* (see "View an Archived Installation Index" on page 394) to display the required installation index revisions.
 2. In the row with the revision for which you want to print the reports, select either or both of the check boxes in the following columns:
 - **Report:** to display the general report for the revision.
 - **Change Report:** to display the change report with the differences between the current and previous revisions.
- TIP** You may only print reports for one revision at a time.
3. Do one of the following:
 - Click .
 - Click **Reports > Print Report**.

Construction Module Revisions

The revision feature is used to keep track of the changes made to an installation index. It is important and useful to have a chronological description of the changes, dates of change, and a list of persons who approved them. Using this feature, you can add revisions in the **Installation Index Manager** window.

1. With the **Installation Index Manager** window open, select the required installation index category from the list.

The list of installation indexes for the selected category is displayed in the lower data window.
2. In the **Report for** field, select the required report (if more than one report is available for the selected category).
3. Do one of the following:
 - Click .
 - Click **Actions > Revision**.
4. Select one of the revision numbering methods (use **P0, P1, P2...** for preliminary revisions or **0, 1, 2 / A, B, C**, and so forth, for normal serial revisions).

TIP When you first select a revision numbering method, several options are available to you, including preliminary revisions (designated by **P0**, **P1**, **P2**...). Once you select one of the other revision methods, you will not be able to return to the preliminary revision method and this option will be disabled.

5. Click **New** to add new revision data.
6. Add the revision data in the appropriate data fields.

TIP The **By** data field contains the current user's initials by default, if previously defined by the SmartPlant Instrumentation System Administrator.

7. When done, click **OK**.

Maintaining a Revision Archive

When you create a new revision for an installation index in the Construction module, a view-only archive copy is made of the information contained in the installation index reports. This information is stored even after the reports themselves have been deleted.

The revision archive data can be saved in the database or in a file, depending on the setting selected in the Administration module.

NOTES

- If you select to save your previous revisions in a file, make sure that: the path to the appropriate file is added to your environment path parameter (see your Windows User Guide to learn about defining paths in your environment).
- If the revision file is shared by a number of users via a local network connection, make sure that the resource on which that file is located is appropriately shared (users should have both read and write access rights to the revision files).
- Identical drive mapping is used for all the appropriate workstations to indicate the location of the psr files.

View an Archived Installation Index

Once you generate revisions for the installation indexes, the revision information is added to the archive, and you can retrieve it when required. You can filter archived indexes according to various criteria (contractor, contract, date ranges, etc.). On retrieval of an archived index, you can edit its revision information and print reports.

1. Do one of the following:
 - Click .
 - Click **Actions > Installation Index Manager Archive**.
2. In the upper part of the window, select the required criteria to filter the retrieved installation indexes, from the following options:
3. In the **Index category** field, select to specify a particular installation index category for filtering the retrieved archived indexes.
4. In the **Report for** field, select the required report (if more than one report is available for the selected category).
5. To specify installation indexes belonging to a particular contract, select the required contract in the **Contract number** field, or leave the field blank to retrieve archived indexes that match the other selection criteria for all the contracts.
6. To specify a particular installation index, select that index in the **Index number** field, or leave the field blank to all archived indexes that match the other selection criteria.

7. To specify installation indexes belonging to a particular contractor, select the required contractor in the **Contractor name** field, or leave the field blank to retrieve archived indexes that match the other selection criteria for all the contractors.
8. Select one of the following options:
 - **Revision:** to specify a particular revision number for retrieval. If you do not specify a revision number, all relevant revisions will be retrieved.
 - **From date / To date:** to specify a range of dates for revisions. If you do not change the defaults (**From date:** 00/00/00, **To date:** <today's date>), all installation index revisions generated up to and including the current date will be retrieved.
9. Do one of the following:
 - Click .
 - Click **Actions > Find**.

NOTE The installation index revisions are retrieved according to the specified selection criteria.

Edit an Archived Revision

If required, edit a revision as follows:

1. Select a revision from the retrieved list.
2. Do one of the following:
 - Click .
 - Click **Actions > Revision**.

TIP When you open the **Revisions** dialog box from the archive, you can only edit existing revisions; you may not create new ones.
3. Edit the revision data as required in the appropriate data fields.
4. When done, click **OK**
You are returned to the **Installation Index Manager Archive** window.
5. To return to the current **Installation Index Manager** window, do one of the following:
 - Click .
 - Click **Actions > Working Installation Index Manager**.

Performing Calibrations

The Calibration module allows you to perform various types of calibrations for your instruments and test equipment, such as functional checks, single-point and multi-point calibrations, fiscal meter proving, and tank level gauging. You can customize your own calibration forms using InfoMaker, and in this way specify which calibration parameters you want to include on the calibration form. Furthermore, you can perform different types of calibrations on a particular instrument by defining separate profiles for each calibration type and associating the desired profiles with the instrument. After you enter calibration data and save the record, the software stores a calibration event in the history repository which includes your data and a visual representation of the calibration form. This way you can monitor the calibration history of each instrument and generate calibration certificates.

You can specify a large variety of calibration parameters such as calibration ranges, trip point values, and alarms. The calibration user interface allows you to easily compare current measured values with the appropriate specifications.

In order to perform calibrations, a calibration infrastructure must exist. This infrastructure includes definitions of profiles, settings and data entry forms, and, if required, the creation of customized calibration forms. For details, see *Flow of Preliminary Activities for Calibration* (see "Calibration Preliminary Activities" on page 403).

IMPORTANT

- If you upgraded the software from a version prior to 2007.5, you must restore the standard forms and regenerate the form pages to be able to use existing calibration settings and data. If you intend to use customized calibration forms only, you may skip this upgrade step. For details, see *Upgrading the Standard Calibration Forms* (on page 400).
- The Calibration module is available only if it is included in the software license that you purchased from Intergraph.

Calibration Module Window

The Calibration module window opens when you click  on the toolbar or click **Module > Calibration**.

The window consists of the following sections:

- **Menu bar** — Provides access to all the *Calibration Module Commands* (on page 396).
- **Application work area** — The region where other Calibration module windows and dialog boxes open.
- **Toolbar** — Provides access to the most common Calibration module commands.
- **Status bar** — Shows the current status or the pertinent Microhelp message, plant hierarchy items, and the current date and time.

Calibration Module Commands

Use the Calibration module commands to access the user interfaces for performing the main calibration tasks.

Icon	Command	Explanation
	Actions > Tag Settings	Opens the Tag Calibration Settings Window for the selected instrument tag.
	Actions > Data Entry	Opens the Calibration Data Entry Window, where you enter calibration data for the selected instrument tag.
	Actions > History	Opens the Calibration History Window, from which you can view the calibration history per tag. The calibration history data includes settings and calibration results.
	Actions > Fluke Interface	Opens the Fluke Interface Wizard to allow you to exchange data between SmartPlant Instrumentation and the Fluke 743B / 744 Documenting Process Calibrators.
	Actions > Calibration Profile Settings	Allows you to select a calibration profile and view or edit the profile settings calibration sheet.

Icon	Command	Explanation
	Reports menu	Allows you to access options for displaying various calibration reports.
	Tables > Calibration Profiles	Opens the Calibration Profiles dialog box, which allows you to define new calibration profiles or modify existing profiles.

Enter Tag Number Dialog Box

This dialog box opens to allow you to enter a tag number for one of the following actions:

- Defining tag calibration settings
- Entering calibration data
- Viewing calibration history

Type the entire tag number in the field, including the correct number of spaces, for example: 101-FE 100 (3 spaces between 'FE' and '100'), or click **Find** to open the **Find Tag** dialog box.

Starting the Calibration Module

Start the Calibration Module from the SmartPlant Instrumentation Window

- Do one of the following:
 - On the **SmartPlant Instrumentation** window menu bar, click **Modules > Calibration**.
 - On the **SmartPlant Instrumentation** window toolbar, click .

Start the Calibration Module from the Instrument Index Standard Browser

1. Open an Instrument Index Standard Browser view.
2. Select instrument tags that require calibration.
3. Do one of the following:
 - Click **Modules > Calibration**.
 - On the main toolbar, click .

Search for Tag Numbers in the Calibration Module

This procedure enables you to find tag numbers in the Calibration module using search parameters in the **Find Tag** dialog box. Entering search parameters enables you to narrow down your search. You can narrow your search to the maximum by entering all the search parameters. It is up to you what search parameters to specify. You can use wildcards in the fields where you type values: underscore (**_**) for single characters, and percent (**%**) for multiple characters.

Leaving some of the search parameter fields empty widens the search. If you do not specify any search parameters, the software finds all the existing tag numbers.

After the search is complete, the software displays the tag numbers that match your search parameters in the **Search results** data window of the **Find Tag** dialog box.

1. With the **Calibration Module** window open, do one of the following:
 - Click **Actions > Tag Settings** to find the desired tag numbers and display them in the **Tag Calibration Settings** window.
 - Click **Actions > Data Entry** to find the desired tag numbers and display them in the **Calibration Data Entry** window.
 - Click **Actions > History** to find the desired tag numbers and display them in the **Calibration History** window.
2. On the **Enter Tag Number** dialog box, click **Find**.
3. If needed, select **Search by category** to find only tag numbers that you have associated with tag categories in the Instrument Index module. For details, see Associate Tag Category and Criticality.

TIPS

- Search tag numbers by category if you have created user-defined tag categories and associated tag numbers with these categories. You can also type a category in the **Category** field to narrow your search to tags associated with this category only. Use wildcards and not case-insensitive characters if needed.
 - When searching for tag numbers by tag category, you can narrow down your search by typing a specific tag category or loop name, or by selecting the required instrument type and process function. Note that you cannot use certain search parameters together with the tag category parameter. These fields become view-only.
4. Define your search criteria using the required combination of the following search parameters:

Search Parameter	Explanation	Example
Tag number	Type the whole tag number you are looking for. Include any prefix, suffix, and separator characters. You can use wildcards if needed.	101-FT 22-25/1 or use a wildcard: 101-FT%
Process function	Select a process function to narrow your search to tags belonging to this process function.	Flow Pressure
Instrument type	Select an instrument type to narrow your search to tags belonging to this instrument type.	D/P Type Flow Element (FE), Mass Flow Transmitter (FT)

Search Parameter	Explanation	Example
Status	Select a tag status to narrow your search to tags associated with this status.	An existing device, a new instrument, a relocated device
Location	Select a tag location, for example: Field , to narrow your search to tags for which you have defined this location.	Equipment room, junction box
I/O Type	Select an I/O type to narrow your search to tags for which you have defined this I/O type.	AO (analog output) DI (Digital input)
Prefix	Type the tag number prefix to find all the tag numbers that contain this prefix in their names. Do not include the separator characters. The unit number segment in the tag number name is usually the tag number prefix. You can also use wildcards if needed.	101
Number	Type the numeric segment of a tag number to find all the tag numbers that contain this numeric segment. You can also use wildcards if needed.	2315
Suffix	Type the suffix segment of the tag number to find all the tags that contain this suffix. Do not type the slash character before the suffix. You can also use wildcards if needed.	1 (the number following the slash (/) in tag number 101-FT -2225/1)
Equipment	Select the equipment to narrow your search to tags for which you have defined this equipment.	
Line	Select a line to narrow your search to tags with which you have associated this line.	

- If desired, select **Look in all units** to look for the defined search criteria in all the units of the current plant.
- Click **Find** to display the tag numbers in the **Search results** data window.
NOTE Select the **Show more search results** check box to hide the search criteria fields and enlarge the **Search results** data window to display more tag rows. Clearing this check box returns to normal view.
- Select the required tag numbers in the **Search results** data window.
TIP You can also select the **Select all** check box to select all the displayed tag numbers.
- Click **OK** to open the appropriate window (**Tag Calibration Settings**, **Calibration Data Entry**, or **Calibration History**) and proceed with the calibration activities.

Calibration Forms

Calibration forms are basis for calibration sheets, which are the means of entering data and viewing results in the Calibration module of SmartPlant Instrumentation. These forms are completely customizable to allow users to specify various types of calibration routines. To customize calibration forms, you must be familiar with InfoMaker. For details, see Calibration Form Customization.

SmartPlant Instrumentation comes shipped with two forms so that existing users can continue to work with legacy data and still see a calibration form layout similar to what existed previously. The names of these forms are **Standard Settings** and **Standard Data Entry**. In addition, there are various shipped .psr files, which you can customize as needed.

You use the Specifications module options to create and manage calibration forms because the form creation mechanism is the same as for specification forms. You can create as many calibration forms as you require; and the forms can differ in type of data, layout and calibration requirements. In the Specifications module, there are two page item types designated for calibration forms: **Calibration Settings** and **Calibration Data Entry**. You must assign one settings form and one data entry form to each calibration profile that you define. There is no specific form for calibration history because calibration history is always based on the data entry form. When opening any window in the Calibration module, you actually open a calibration form assigned to a calibration profile.

NOTES

- Any calibration form, whether shipped or user-defined, can only comprise a single data page.
- A calibration form page may include a title block in the body of the page, as one the page portions. However, in calibration form pages, you do not create revisions or revision data.
- If you upgraded the software from a version prior to 2007.5, you must restore the standard forms and regenerate the form pages to be able to use existing calibration settings and data. If you intend to use customized calibration forms only, you may skip this upgrade step. For details, see *Upgrading the Standard Calibration Forms* (on page 400).

See Also

Flow of Preliminary Activities for Calibration (see "Calibration Preliminary Activities" on page 403)

Upgrading the Standard Calibration Forms

When upgrading the software from a version prior to 2007.5, the setup makes changes that allow you to continue to use existing calibration settings and data if desired. Several of the former calibration types have been combined into a single multi-point calibration. All existing tag settings and history event records will be updated accordingly. In addition, some new calibration types have been added. The following table shows the differences between the available calibration types before and after the upgrade.

Before Upgrade	After Upgrade	Description
Other, Single point, Three point, Five point, Five point calibration and hysteresis, Eleven point	Multiple points	Allows you to specify any number of points for calibration. See Notes below for limitation on the number of points.
Function check	Function check	Does not specify any calibration values. This option is used to check whether or not the instrument or equipment is working.

Switch set point	Switch set point	Allows you to specify up to four calibration points for a temperature or pressure switch.
Alarm/Trip/Set point	Alarm/Trip/Set point	Specifies a single measurement point for calibrating an alarm, a trip, or a set-point.
—	Fiscal flow meter proving	A type of calibration for determining meter factor and repeatability. Requires you to customize your own forms and profiles if you want to use this option.
—	Tank level gauging	A type of calibration for tank- level gauging. Requires you to customize your own forms and profiles if you want to use this option.

NOTES

- To ensure continued support for existing tag settings, the program creates a standard (shipped) profile with two associated forms: **Standard Settings** and **Standard Data Entry**.
- The standard profile is associated with each existing instrument type.
- Each existing tag setting is associated with the **Standard Settings** form.
- Each existing calibration result record is associated with the **Standard Data Entry** form.
- The **Standard Settings** and **Standard Data Entry** forms support up to 11 calibration points for As-Found or As-Left data. If you want to enable more than 11 points, you must customize the forms using InfoMaker.
- New properties are added to the calibration history points for the As-Found and As-Left expected values and calculated errors.
- The software creates a snap-shot of all former calibration history records and points based on the **Standard Data Entry** form (this also includes results that were previously collected by a Fluke Calibrator).
- Additional units of measure properties are available for trip points in the tag calibration settings and data entry. Previously, a single unit of measure property was used. The change should be transparent for users. The upgrade process will populate these properties with the necessary unit of measure value that was used.
- During the upgrade, test equipment items that were populated in the **Test Equipment** table are re-created as tag numbers with the tag class **Test Equipment**. The test equipment name, description, and serial number are copied to the database (the serial number can only be displayed in a browser view and not on the **Tag Number Properties** dialog box for the test equipment). Other test equipment properties (**Test Date**, **Tested By**, **Accuracy**, and **Units of Measure**) are not transferred as a result of the upgrade.
- During the upgrade, many other new properties are added to the calibration tables to support new functionality.
- The fields **Compnt_Mfr_ID** and **Compnt_Mod_ID** display the current manufacturer and model name associated with the tag. The fields **Compnt_Mfr_Name** and **Compnt_Mod_Name** display the history of changes made to the manufacturer and model name when saving the **Calibration Data Entry** sheet.

Upgrade the Standard Calibration Forms

You must perform this procedure once after the upgrade to prepare the new tag calibration settings and data entry forms so that they can use existing and new data.

1. In the **Specifications Module** window, click **Edit > Form Editor**.
2. Do the following to restore the **Standard Settings** calibration form:
 - a. In the **Form Editor**, click **Actions > Restore Form**.
 - b. Select the **Standard Settings** form and click **OK** to restore that form.
3. Repeat the above steps to restore the **Standard Data Entry** calibration form.
4. Close the **Form Editor** window.
5. Click **File > Preferences > Specifications**, and on the **General** tab, under **Calibration profile forms**, select **Reuse columns in form page**.
6. In the **Specifications Module** window, click **Edit > Page Editor**.
7. Do the following to regenerate the **Standard Settings** calibration form page:
 - a. In the **Page Editor**, click **Actions > Open Page**.
 - b. On the **Select Specification Page** dialog box, select the **Standard Settings** form page and click **OK**.
 - c. Click **Actions > Regenerate Page**.
 - d. On the **Regenerate Page** dialog box, under **Select tables, views and columns**, select a check box for each table or view from which you want to display additional columns in the **Table Column List** pop-up window.

NOTES

- Some tables, such as Component and Flow, do not appear in the list. These tables contain mandatory fields and are also regenerated. The fields will appear in the **Table Column List** window when you open **Edit > Column List**.
 - For the standard form pages, the tables are already selected.
 - If you have not made any changes to the page, skip this step and the two following steps.
- e. Under **Show Columns**, click **Show** for each required table or view.
 - f. Under **Select columns**, do one of the following:
 - Select the **Select all** check box to select all the columns in the table.
 - Hold down the **Ctrl** key and select or clear specific columns as desired.
 - g. Click **OK** to regenerate the page.
 - h. Click **Save** .
8. Repeat the above steps to regenerate the **Standard Data Entry** form page.
 9. If you have existing calibration records that you want to save as history events in the current version, do the following:
 - a. Run the CalibrationResultsUpgrade.exe file, which is located in the SmartPlant Instrumentation home folder.
 - b. On the **Calibration Results Upgrade Utility** dialog box, select an appropriate domain.
 - c. If you do not want to keep your previous results, select the **Overwrite existing results** check box.

TIP If you have updated your calibration forms, this step will ensure that the software re-creates previously saved history events for the updated forms.

- d. Click **OK**.

NOTE If you have previous calibration results that originated from a Fluke Calibrator, and you want to save history events for these results using the Fluke forms, first save the Fluke forms as standard calibration forms before running the CalibrationResultsUpgrade.exe utility. This results in the software saving all of your existing calibration data as history events using the Fluke forms. For details of how to use the Fluke forms, see *Set up a Profile for the Fluke Calibrator* (on page 421).

Upgrade Calibration Results

After upgrading a database with calibration results that were obtained in a version prior to 2007.5, you need to run the calibration results upgrade utility so that your existing calibration results become compatible with the new method of displaying calibration history.

1. Run the CalibrationResultsUpgrade.exe file, which is located in the SmartPlant Instrumentation home folder.
2. On the **Calibration Results Upgrade Utility** dialog box, select an appropriate domain.
3. If you do not want to keep your previous results, select the **Overwrite existing results** check box.
4. Click **OK**.

IMPORTANT If you are upgrading custom calibration forms to version 2009, after the upgrade, you need to regenerate them in the Calibration module.

Calibration Preliminary Activities

This topic describes the flow of activities for creating and customizing calibration forms, assigning them to calibration profiles, and associating the profiles with appropriate instrument types. These activities are necessary before you can start defining calibration settings for instruments.

IMPORTANT If you upgraded the software from a version prior to 2007.5, you must restore the standard forms and regenerate the form pages to be able to use existing calibration settings and data. If you intend to use customized calibration forms only, you may skip this upgrade step. For details, see *Upgrading the Standard Calibration Forms* (on page 400).

1. Customize calibration form pages in InfoMaker

SmartPlant Instrumentation comes shipped with a number of .psr files that you can customize according to your needs using InfoMaker. For details, see *Customizing Form Pages in InfoMaker Common Tasks*.

2. Add the customized forms to the database

Using the Specifications module, you need to retrieve the customized form pages from the .psr files, and associate the pages with the **Calibration Settings** or **Calibration Data Entry** form types, and then save the forms to the database. For details, see *Add Customized Calibration Forms to the Database* (on page 404).

3. Create calibration profiles

Using the Calibration module options, you must now create calibration profiles and assign the customized forms to the desired profiles. You must assign one calibration settings and one calibration data entry form to each profile. For details, see *Create a Calibration Profile* (on page 405).

4. Define calibration profile settings

After creating your calibration profile and assigning forms to the profile, you use the Calibration module options to enter values in the **Calibration Profile Settings** window. These values then appear in the **Tag Calibration Settings** window for the particular instrument or test equipment tag for which you select that profile. For details, see *Define Calibration Profile Settings* (on page 405).

5. Associate the calibration profiles with instrument types

Using the Instrument Index module options, you need to associate one or more profiles with the appropriate instrument types. You can associate several profiles with a specific instrument type and, if needed, set one profile as default. Also, it is possible to associate the same profile with several instrument types. For details, see *Associate Calibration Profiles with Instrument Types* (on page 405).

6. Managing code tables and test equipment

In order to perform certain calibration activities, you can specify test equipment for checking the accuracy of the calibration values and you can also set up various codes for classifying your calibration results. For details about test equipment, see *Test Equipment* (see "Defining Test Equipment" on page 407). For details about various codes, see *Manage Code Tables* (on page 406).

Add Customized Calibration Forms to the Database

1. In the **Specifications Module** window, click **Edit > Page Editor**.
2. On the **Page Editor** menu bar, click **Actions > Open File**:
3. On the **Open File** dialog box, navigate to the folder where your .psr file is located and select the file.
4. Click **OK** to open the page in the **Page Editor**.
5. Regenerate the page. For details, see *Regenerate a Specification Page*.
6. Save the page and close the **Page Editor**.
7. In the **Specifications Module** window, click **Edit > Form Editor**.
8. On the **Form Editor** menu bar, click **Actions > New Form**.
9. On the **Select Specification Page** dialog box, do one of the following:
 - To create a calibration settings form, select the page whose page item type is **Calibration Settings**.
 - To create a calibration data entry form, select the page whose page item type is **Calibration Data Entry**.
10. Save the form and close the **Form Editor**.

TIP Calibration forms are single-page forms; therefore, it is not possible to add pages to forms whose page item type is **Calibration Settings** or **Calibration Data Entry**.

TIP When saving your form, make sure that the name of the form is unique, to prevent you from overwriting previous forms.

Create a Calibration Profile

1. Open the Calibration module.
2. Click **Tables > Calibration Profiles**.
3. On the **Calibration Profiles** dialog box, from the **Calibration type** list, select the desired calibration type for the profile you are going to create.
4. Click **New**.
5. In the **Calibration Profile** column, type a name for the new calibration profile.
IMPORTANT All calibration profile names must be unique, regardless of the calibration type to which the profile belongs.
6. Type a description and a procedure if desired.
7. Click **Assign Forms**.
8. On the **Assign Forms** dialog box, select the desired calibration settings and calibration data entry forms and click **OK** to return to the **Calibration Profiles** dialog box.
9. Click **OK** to close the **Calibration Profiles** dialog box.

See Also

Calibration Forms (on page 400)

Define Calibration Profile Settings

1. Open the Calibration module.
2. Do one of the following:
 - Click **Actions > Calibration Profile Settings**.
 - Click .
3. On the **Select Calibration Profile** dialog box, from the **Select calibration type** list, select the calibration type to which the profile you want to edit belongs.
4. Select the desired calibration profile.
5. In the **Calibration Profile Settings** window, enter the desired values for the profile properties (these will usually define typical output range, number of points, and tolerances).
6. When done, save the profile settings.

See Also

Define Tag Calibration Settings (on page 409)

Associate Calibration Profiles with Instrument Types

IMPORTANT Before associating calibration profiles, you must create them in the Calibration module. For details, see *Create a Calibration Profile* (on page 405).

1. In the Instrument Index module, open the **Instrument Type Profile** dialog box for the target instrument type. For details, see *Define an Instrument Type Profile*.
2. Click the **Calibration** tab.
3. Click **Associate**.
4. On the **Associate Calibration Profile** dialog box, do the following:
 - a. Select a calibration type.
 - b. Select a calibration profile.
 - c. Click **OK**.

NOTE It is only possible to select and associate one profile at a time.

5. Repeat steps 3 and 4 for other profiles you want to associate with the current instrument type.
6. If you want to set one profile as default, on the **Calibration** tab, in the **Default** column, select a check box beside the target profile.

NOTE If you set a profile as default, when opening calibration settings for a new instrument, you bypass the **Select Calibration Profile** dialog box and automatically open the **Tag Calibration Settings** window. In the **Tag Calibration Settings** window, the software loads the **Calibration Settings** form assigned to the default profile, including data values you defined in the **Calibration Profile Settings** window. You do not have to set a profile as default. If more than one profile is available for the instrument type, and none of them are marked as default, the **Select Calibration Profile** dialog box opens, where you select the profile you need, and then open the calibration settings.

7. On the **Instrument Type Profile** dialog box, click **Apply** or **OK**.

Dissociate Profiles from Instrument Types

1. In the Instrument Index module, open the **Instrument Type Profile** dialog box for the target instrument type. For details, see Define an Instrument Type Profile.
2. Click the **Calibration** tab.
3. Select a profile and click **Dissociate**.

NOTES

- You can dissociate only one profile at a time.
 - You can dissociate any profile, including profiles you already used for creation of calibration settings. Once calibration settings are created for a given instrument, they are saved in the database for that instrument.
 - If more than one profile is available for the instrument type, and you dissociate the default profile without defining a new one, when opening calibration settings for a new instrument, you can no longer open the **Tag Calibration Settings** window automatically. If no profile is set as default, the software first opens the **Select Calibration Profile** dialog box, where you select the profile you need, and then you can open the **Tag Calibration Settings** window.
4. On the **Instrument Type Profile** dialog box, click **Apply** or **OK**.

Manage Code Tables

There are four code tables that you can use to specify your own set of values that you can use for classifying your calibration results: **Action Codes**, **Damage Codes**, **Diagnostic Codes**, and **Result Codes**. After you define your own codes in each of these tables, the values become available for selection on the **Standard Data Entry** calibration form or on any customized calibration form where you choose to display the codes.

NOTE There are three predefined result codes: **Passed**, **Failed**, and **None** that you cannot edit or delete from the database. These values do not appear in the **Result Codes** dialog box. An additional value, **Bad Result**, is also available when editing history results.

1. In the **Calibration Data Entry** window, on the **Tables** menu, click the desired code table.
2. On the dialog box that opens, do one of the following:
 - Click **New** to add a new value.
 - Highlight an existing value.

3. Type the desired code and its description in the appropriate columns.

See Also

Enter Calibration Data (on page 412)

Defining Test Equipment

SmartPlant Instrumentation allows you to define and assign test equipment to the instruments in your plant. Test equipment is used to check that the calibration of your plant's instruments is accurate and operating within their given parameters, and to recalibrate those instruments that do not meet their required parameters. You can assign one item of test equipment to many instruments. Each piece of test equipment has assigned to it one or more user defined profiles. A profile is a template of the default settings/values used by the test equipment. For example, checking the minimum/maximum limits of various temperature gauges, you can assign a different profile to your test equipment, that represents the calibration requirements, for each temperature gauge. Test equipment tags are maintained on the domain level in the Test Equipment folder and can only be assigned to As-Built plants.

NOTES

- If you are using the Fluke Documenting Process Calibrator models 743B or 744, there is a separate interface and special forms that you use for exchanging and displaying the calibration data. For details, see *Working with the Fluke Interface* (on page 420).
- Test equipment cannot include the following data:
 - I/O types
 - P&IDs
 - Process equipment
 - Process lines
 - Process data
 - Electrical power requirements (SmartPlant Electrical integration)
 - Associated symbols for the Enhanced Report Utility
 - Calculation module data
 - Wiring data
 - Loop data or association

See Also

Selecting Test Equipment (see "Select Test Equipment" on page 413)

Prerequisites for Creating a Test Equipment Tag

The following prerequisites are necessary for creating a test equipment tag.

A free segment text name must be created in the Administration module. For easy identification of a specific item of test equipment, when creating a naming convention for test equipment it is advisable to include the serial number of the equipment as part of the tag. For more details, see **Define Naming Conventions** in the *Administration User's Guide*, under *Domain and Project Administration > Naming Conventions*.

1. Define the **Instrument Type Profile** for the specific test equipment.

NOTE When creating the **Instrument Type Profile** you must select **General** as the process function and **Test Equipment** in the **General Process Function Sub-Category** column of the data window, then in the **Instrument Type Profile** dialog box you can associate a **Specification Form** with the profile. For more details, see *Define an Instrument Type Profile*.

2. Create the **Test Equipment** tag. For more details, see *Create a Test Equipment Tag* (on page 408).
3. On the calibration sheet, enter calibration data and choose the desired test equipment. For more details on calibration, see *Performing Calibrations* (on page 395).

Create a Test Equipment Tag

1. In the Domain Explorer, do one of the following:
 - In the project where you want to add the **Test Equipment** tag, locate the **Instruments** folder.
 - Locate the **Test Equipment** folder
2. Right-click, and from the shortcut menu click **New > Test Equipment**.
3. On the **New Tag Number** dialog box, enter the new tag number in the **Tag Number** field and click **OK**.
4. On the **Select Test Equipment Type** dialog box, select the required test equipment.
5. On the **Tag Number Properties** dialog box, enter data as required.

The new tag is created in the **Test Equipment** folder located at the domain level of the **Domain Explorer**.

See Also

Prerequisites for Creating a Test Equipment Tag (on page 407)

Test Equipment (see "Defining Test Equipment" on page 407)

Selecting Test Equipment (see "Select Test Equipment" on page 413)

Calibration Settings Activities

This topic describes the main flow of activities when defining settings for calibrating instruments.

IMPORTANT

- If you upgraded the software from a version prior to 2007.5, you must restore the standard forms and regenerate the form pages to be able to use existing calibration settings and data. If you intend to use customized calibration forms only, you may skip this upgrade step. For details, see *Upgrading the Standard Calibration Forms* (on page 400).
- In order to perform calibrations, a calibration infrastructure must exist. This infrastructure includes definitions of profiles, settings and data entry forms, and, if required, the creation of customized calibration forms. For details, see *Flow of Preliminary Activities for Calibration* (see "Calibration Preliminary Activities" on page 403).

1. Create a tag number and define the tag calibration settings

When you create a tag number for an instrument or test equipment, the instrument type of that tag number must be associated with one or more calibration profiles suitable for the type of calibration to be performed. When opening the tag calibration settings in the Calibration module, you must select the desired calibration profile. The software loads the values from that profile into the tag calibration settings, which you can then view and edit in the **Tag Calibration Settings** window. If you select a different calibration profile, the software loads settings values from that profile to the tag calibration settings and overwrites any existing values. The tag calibration settings determine what kind of input data you require for your calibration, such as ranges, input and output units of measure, and tolerances. These settings serve as a reference for any calibrations you perform on the tag provided you do not modify them or select a different profile when performing data entry. For details, see *Define Tag Calibration Settings* (on page 409).

2. Copy existing process data

If you are defining calibration settings for a tag number associated with process data, you use this procedure to copy existing maximum and minimum process data range values into the **Variable Min** and **Variable Max** fields; you can also copy existing alarm and trip data into the appropriate **Alarm / Trip** fields. For details, see *Copy Existing Process Data* (on page 410).

Define Tag Calibration Settings

IMPORTANT Only one set of tag calibration settings is allowed per tag. Therefore you should use the tag calibration settings to define tag-specific data such as alarm / trip points, input and output ranges, and units of measure. Other calibration values that are not tag-specific such as number of calibration points, tolerance, and so forth, can be stored in the calibration profile settings.

1. Start the Calibration module.
2. Do one of the following:
 - Click **Actions > Tag Settings**.
 - Click .
3. On the **Enter Tag Number** dialog box, type the desired tag number or click **Find** to search instrument tags.

NOTE If you selected multiple tag numbers, click **Actions > Tag List** and then click a tag number to define the settings for that instrument.

4. Click **Actions > Select Profile**.
5. Define the calibration settings values for the tags as desired.

TIPS

- If you are defining calibration settings for a tag number associated with process data, you can copy existing process data minimum and maximum range values. For details, see *Copy Existing Process Data* (on page 410).
- If you are using the **Standard Settings** calibration sheet, the **Measuring Point** parameter is available for single point calibrations only.
- If you are using the **Standard Settings** calibration sheet, the fields in the **Set values** group box are available only if under **Calibration type**, you selected **Switch set point**.

6. Click **Save**  to save the settings.

See Also

Enter Calibration Data (on page 412)

Define Calibration Profile Settings (on page 405)

Copy Existing Process Data

1. Open the desired instrument tag in the **Tag Calibration Settings** window.

IMPORTANT

- It is not possible to copy process data for control valves.
 - Process data does not apply to test equipment.
2. Click **Options > Copy Process Data**.
 3. On the **Copy Process Data** dialog box, click **OK**.

NOTES

- You can edit the data in the Process Data module only.
- The copied data overwrites your previous settings in the **Tag Calibration Settings** window.
- If you defined multiple process data cases for the current tag, the data displayed in the **Process Data** pop-up window is determined by the governing case. For details, see *Multiple Process Data Cases*.
- The software copies the **Min** and **Max** values beside **Required range** to the **Variable minimum** and **Variable maximum** fields as well as the alarm and trip data.

Define Alarm / Trip Settings

You enter the required alarm / trip settings in the appropriate **Point** data window. You can specify up to six trip or alarm indication points. For each point, you select a unit of measure, enter the point values and direction, and type a brief description.

1. Open the **Tag Calibration Settings** window for an instrument tag.
2. In the **Alarm / Trips** section, for **First Point**, type the desired values or select them from the lists. When done, define other calibration points as needed.

TIP

If you are defining calibration settings for a tag number associated with process data, you can copy existing process data minimum and maximum range values. For details, see *Copy Existing Process Data* (on page 410).

3. Click **Save**  to save all your calibration settings.

Define the Set Point

After defining the alarm and trip settings, you can also define the set point of the selected instrument. You can either define the set point to be identical to one of the alarm / trip points that you have already defined, or you can define the set point manually.

1. Open the **Tag Calibration Settings** window for an instrument tag.
2. In the **Set Point** section, from the **Point** list, select one of the following:
 - **1st Point, 2nd Point, 3rd Point**, and so forth (any defined calibration point up to the 6th point): The set point will be identical to the selected alarm/trip point and the rest of the fields in this data window become unavailable. Note that this option is possible only after making your alarm/trip definitions.
 - **Free**: You can define the set point yourself.
 - **None**: You can make no set point definitions.
3. Type the rest of the values as desired or select them from the lists.
4. Click **Save**  to save all your calibration settings.

Calibration Data Entry Activities

This topic describes the main flow of activities when entering calibration data for instruments with predefined calibration settings.

1. Select a profile

Profiles provide a means of allowing you to perform different types of calibration for the same tag. When you select a profile in Data Entry mode, the software loads the profile values into the **Calibration Data Entry** form. For details, see *Select a Calibration Profile* (on page 412).

NOTE If desired, you can by-pass profile selection by choosing the existing tag settings.

2. Enter calibration data

After selecting a profile or the existing settings for your tag, you enter the calibration data. The software uses any formulas you have defined on the Data Entry form to calculate calibration results. While entering data, the calibration result is accepted as long as the input values do not exceed the specified tolerances which are set by the expressions on the form. On saving a calibration record, the software generates a history event. For details, see *Enter Calibration Data* (on page 412).

3. View calibration history

After you enter calibration data and save the record, the software stores a calibration event in the history repository which includes your data and a visual representation of the calibration form. You can select desired history events for a particular tag and view and compare an instrument's calibration record. You can also print calibration history reports. Some properties may still be editable if so defined on the **Calibration Data Entry** form; these properties are related to the **Codes** section on the **Standard Data Entry** form. For more information, see *View Calibration History* (on page 414).

4. Generate calibration reports

Various calibration reports are available for viewing and printing. These include a Calibration Check List, Calibration Data, various reports that allow you to analyze failure results, and Calibration History. For details, see *Generate Calibration Reports* (on page 416).

Select a Calibration Profile

1. From the **Calibration Data Entry** window, open data entry sheets for one or more instrument tags and do one of the following:
 - Click **Actions > Select Calibration Profile**.
 - Click .
2. On the **Select Calibration Profile** dialog box, clear the **Use data from tag calibration settings** check box.

CAUTION Loading a profile will overwrite all settings data other than the calibration ranges.
3. Under **Select calibration profile associated with the instrument type**, select the desired calibration profile.

TIP To by-pass the profile and to use the values directly from the instrument tag calibration settings, select the **Use data from tag calibration settings** check box.

NOTES

- If you selected multiple instruments, the software will open the calibration data entry sheet for each instrument in turn and copy data from the selected calibration profile to the data entry fields.
- When copying data from a profile, the software does not change the values for the tag calibration settings.

See Also

Create a Calibration Profile (on page 405)

Define Calibration Profile Settings (on page 405)

Enter Calibration Data

1. In the **Tag Calibration Settings** window, do one of the following:
 - Click **Actions > Data Entry**.
 - On the module toolbar, click .
2. Finish entering the tag's calibration settings and click **Save** . You are prompted to open the **Calibration Data Entry** window for the current tag.
3. In the **Calibration Data Entry** window, on the module toolbar, click **Refresh** .
4. Click the **Data Entry** tab and enter values in each section by typing or selecting from the lists.
5. Do the following for each of the points that you defined for the current tag in the **Tag Calibration Settings** window:
6. In the **Calibration Data Entry** window, in the **Alarms / Trips** section, click **First Point**, **Second Point**, **Third Point**, **Fourth Point**, **Fifth Point**, or **Sixth Point**.
7. Under **As Found**, type the value before calibration.
8. Under **As left**, type the value after calibration.
9. In the **Codes** section, enter general calibration information as you require.
10. From the **Result Code** list, select the desired predefined result code, or custom result code predefined in the **Result Code** dialog box.
11. If from the **Result code** list you selected **Failed** or any other option that indicates a calibration result failure, from the **Diagnostic code** list, select a calibration diagnostic code for this kind of failure.

12. If from the **Result code** list you selected **Failed** or any other option that indicates a calibration result failure, from the **Action code** list, select an action code associated with a solution description for this kind of failure.

13. If from the **Result code** list you selected **Failed** or any other option that indicates a calibration result failure, from the **Damage code** list, select a damage code associated with a solution description for this kind of failure.

TIP If a desired **Result**, **Diagnostic**, **Damage**, or **Action** code is not available from the list, you can define additional codes by selecting the appropriate item on the **Options** menu.

14. Enter additional values, and a short note if needed.

15. Type the pre-calibration values in the **As Found** column.

NOTE On the standard calibration data entry page, error values that exceed the maximum permissible error are highlighted in red, while values that exceed the intermediate error tolerances but do not exceed the permissible error are highlighted in light blue.

16. After calibration of the instrument, type the values in the **As Left** column.

NOTES

- Any error values still highlighted in red show that this instrument's calibration failed. When you save the updated data, the software prompts you to create a Work Request. For details, see *Performing Breakdown Maintenance* (on page 359).
- To enter data for custom fields, click the **Custom** tab.
- The Domain Administrator can enable or disable the use of custom fields. Note that the custom fields in the **Tag Calibration Settings** window are independent of the custom fields in the **Calibration Data Entry** window.

17. Click **Save** .

18. To browse through the tags that you selected, click  or .

NOTES

- Errors are calculated to four decimal places. Any error smaller than 0.0001 will be displayed as zero (0) and stored in the database as zero.
- The fields **Compnt_Mfr_ID** and **Compnt_Mod_ID** display the current manufacturer and model name associated with the tag. The fields **Compnt_Mfr_Name** and **Compnt_Mod_Name** display the history of changes made to the manufacturer and model name when saving the **Calibration Data Entry** sheet.

See Also

Calibration Error Calculations (on page 417)

Define Tag Calibration Settings (on page 409)

Select Test Equipment

On the **Data Entry** sheet for a particular instrument tag, you may specify up to four fields for selecting test equipment items. If desired, you can select the same test equipment item more than once (you may need to do so when using the same equipment for different calibration purposes). Then, in the **Calibration History** window, you can view detailed test equipment information for each calibration entry. Detailed test equipment information also appears in the Instrument Calibration Results report.

View Calibration History

1. Start the Calibration module.
2. Do one of the following:
 - Click **Actions > History**.
 - Click .
3. On the **Enter Tag Number** dialog box, type the desired tag number or click **Find** to search for tag numbers.

TIP If you selected multiple tag numbers, click **Actions > Tag List** and then click a tag number to view the history event for that instrument.
4. Open the desired instrument tag in the **Calibration History** window.
5. If more than one history record exists for the selected instrument tag, click **Previous**  or **Next**  to navigate to the desired record.

NOTE The fields **Compnt_Mfr_ID** and **Compnt_Mod_ID** display the current manufacturer and model name associated with the tag. The fields **Compnt_Mfr_Name** and **Compnt_Mod_Name** display the history of changes made to the manufacturer and model name when saving the **Calibration Data Entry** sheet.

List of Calibration Reports

The following list describes some of the reports that are available in the Calibration module.

Report Name	Description
Instrument List	Displays a list of the available instruments organized by system category. Instruments that are associated with more than one category appear once under each category.
Instrument Structure	Displays a list of the available instruments organized by the loops to which they belong.
Calibration Codes — By Instrument Type	For the selected code category, displays the number of instruments for which each particular code value applies, organized by instrument type.
Calibration Codes — Monthly Analysis	For the selected code category, displays a month by month analysis over the specified date range of the number of instruments for which each particular code value applies, organized per loop.
Result Code — Result Analysis	Displays for the specified date range an analysis of the instruments per result code. Note that the result code is a user-defined property that has nothing to do with the Calibration result property, which the software calculates, and which can have the value Passed or Failed .
Check List — By Loop	For one or more selected loops, displays a check list for each instrument showing principal settings and input data used for calibration. You can print this report and write As Found, As Left, and code values as needed.

Report Name	Description
Check List — By Tag	For the instruments selected using the filter, displays a check list showing principal settings and input data used for calibration. You can print this report and write As Found, As Left, and code values as needed.
History Event Details	For one or more selected loops, displays a brief summary of the calibration history of all the instruments in those loops, including loop errors where relevant.
Calibration Results	For the instruments selected using the filter that have either passed or failed the calibration, displays a summary of each instrument's calibration settings and results.
Calibration Equipment	Allows you to specify one or more items of calibration equipment and displays the frequency of calibration failure for each item of calibration equipment. You can then analyze the results to find out if there is a fault in the calibration equipment itself.
Loop Error	For one or more selected loops for which the loop error was calculated, displays the error variation over the specified date range at the specified calculated point. This report is useful for checking the zero drift of a particular loop over time.
Component Error	For one or more selected instruments, displays the error variation over the specified date range at the specified calculated point. This report is useful \ for checking the zero drift of an instrument over time.
Alarm / Trip Points Summary	Generates a summary of the trip and alarm point data of all selected tags.
Calibration Certificate (available from main Calibration module window only)	Generates a summary of the selected tag's calibration results.
Calibration Results Report	Generates a summary of the selected tag's calibration results, calibration settings, a graphic representation of the error percentage, and test equipment used for calibrating this tag.

Generate Calibration Reports

1. In the appropriate Calibration module window, on the **Reports** menu, click the menu option for the report that you want to generate.
2. If a filter dialog box opens, do the following, as available:
3. In the **Date range** group box, do one of the following to set the date range for the report:
 - To set a specific date range, select **Specified dates**, and then in the **From** and **To** lists enter the date range.
 - To generate a report for the [previous, current, or next] [week, month, or year], select **Predefined period**, and then in the lists to the right, select the options that you need.
4. In the **Filter by** group box, use the options to set a filter condition for the report that you are generating.
5. Click **OK**.

NOTES

- For reports by tag or by loop, click  to search for the items that you want.
- The loop error report can be displayed even if a loop error calculation was not performed on that day; however, the report is only of an earlier calculation date. The default calculation method that the report depends upon is Normal values. However, if you select a calculation method other than Normal for a certain loop the calculation report changes respectively.
- To generate a calibration equipment report, type the name of the equipment, with or without wildcards, in the **Calibration equipment** field. If you do not enter a value for calibration equipment, you can click **OK** and select the desired item from the **Select Test Equipment** dialog box.
- The fields **Compnt_Mfr_ID** and **Compnt_Mod_ID** display the current manufacturer and model name associated with the tag. The fields **Compnt_Mfr_Name** and **Compnt_Mod_Name** display the history of changes made to the manufacturer and model name when saving the **Calibration Data Entry** sheet.

Loop Error Calculation

The loop error feature offers you three different methods to calculate the total error of a selected loop: normal, absolute, and root mean square. The calculation is performed per loop based on one or more instrument tags belonging to the loop.

The error is measured at several points that you predefine in the **Tag Calibration Settings** window. The error is calculated at the default point that you define in the **Preferences** dialog box. The loop error is measured relative to the largest tolerance value of all the calibration data for the tags that are included in the loop error calculation.

IMPORTANT

- To make a valid loop error calculation, make sure that your calibration and calculation have been time stamped on the same day.
- Set the value of the loop error calculation method and the loop calculation point on the **Preferences** dialog box. For details, see Calibration and Maintenance > General (Preferences).
- Define the calibration settings for the selected tags. Tags have their predefined settings which you enter in the **Tag Calibration Settings** window. When you calculate the loop error of a specific loop you should use the settings to include at least one tag that belongs to the designated loop. For details, see Setting Tags for Loop Error Calculation.

- Enter calibration data for the selected tags in the loop. For details, see *Enter Calibration Data* (on page 412).
- Select the desired calculation method and perform the actual calculation. For details, see *Calculating Loop Error*.
- Generate a loop error report. For details, see *Generate Calibration Reports* (on page 416).

Calibration Error Calculations

This topic describes how the software calculates the calibration errors.

As Found and As Left Errors (% of Span) Calculation

The following formula is used to calculate the **As Found** and the **As Left** errors (% of **Span**):

$$AF / AL \% \text{ of Span} = \frac{AF_i - ENG_i}{OUT\ SIG_{max} - OUT\ SIG_{min}} \times 100\%$$

Where i = calibration point number

As Found and As Left Errors (% of Reading) Calculation

The following formula is used to calculate the **As Found** and the **As Left** errors (% of **Reading**):

$$AF / AL \% \text{ of Reading} = \frac{AF_i - ENG_i}{ENG_i} \times 100\%$$

Where i = calibration point number

Error Calculation of Square Root Extraction

Calibration errors for differential pressure flow instruments that require a square root extraction are calculated according to the following formula:

$$OUTPUT = OUT.SIGNAL_{min} + \left[OUT.SIGNAL\ RANGE * \sqrt{\% \text{ OF INPUT}} \right]$$

Example:

`out.signalmin = 4 mA`

`out.signalmax = 20 mA`

`out.signal range = out.signalmax - out.signalmin = 16mA`

% of Input	Output Value
0	$4 + (16 * 0) = 4$
25%	$4 + (16 * 0.5) = 12$
50%	$4 + (16 * 0.707) = 15.3137$
75%	$4 + (16 * 0.866) = 17.8564$
100%	$4 + (16 * 1) = 20$

NOTE Errors are calculated to four decimal places. Any error smaller than 0.0001 will be displayed as zero (0) and stored in the database as zero.

Set Tags for Loop Error Calculation

This procedure explains how to set tags associated with a certain loop for inclusion in the loop error calculation.

IMPORTANT

- You must calibrate the tags before you perform the actual loop error calculation.

You can also set the tags for inclusion from a view that you create in the Browser module. For details of how to create a view, see [Add a New View](#). If you use a browser view to set the tags, make sure that you include the **Tag Number**, **Loop Number**, and **Calibration Required** columns.

1. Click **File > Preferences**.
2. On the **Preferences** dialog box, do the following:
 - a. In the tree view, click **Calibration > General**.
 - b. From the **Calculation method** list, select the desired loop calculation method. Select **Ask user** if you want the software to prompt the user for the method at the time of performing the loop error calculation.
 - c. Beside **Calculate at point**, enter a value between 1 and 100.

TIP The value is set by default to 75%.

3. Start the Calibration module.
4. Do one of the following:
 - Click **Actions > Tag Settings**.
 - Click .
5. On the **Enter Tag Number** dialog box, type a tag number or click **Find** to find the required instrument tags.

NOTES

- You must select at least one tag for each loop that you want to calculate.
 - If you selected more than one tag, click **Actions > Tag List** and then click the desired tag number to define the settings for that instrument.
6. In the **Tag Calibration Settings** window, in the **Errors / Points** section, from the **Include in loop error** list, select **Yes**.
 7. Define all the remaining settings as needed.
 8. Click **Save**  to save the settings.

NOTE You can now start entering calibration data by selecting **Yes** when prompted after saving the data. Click **No** when prompted to start the data entry to continue to define calibration settings for another tag number.

Calculate Loop Errors

After you set tags for loop error calculation, use this procedure to do the following:

- Select tags from a given loop for loop error calculation
 - Select a loop error calculation method
 - Perform the loop error calculation
1. In the **Calibration Module** window, do one of the following:
 - Click .
 - Click **Actions > Data Entry**.
 2. On the **Enter Tag Number** dialog box, click **Find** to open the **Find Tag** dialog box.
 3. Under **Search parameters**, set conditions to filter the tag display as necessary and click **OK**.

IMPORTANT

- To ensure that an instrument tag is included in a loop error calculation, in the **Tag Calibration Settings** window, you must set **Include in loop error** for that tag to **Yes**. For details, see *Setting Tags for Loop Error Calculation*.
 - For all instrument tags that are included in a loop error calculation, you must specify the **Tolerance units of measure** value as %.
4. Under **Search results**, select the tags that you want to include in the loop error calculation, and click **OK**.
 5. In the **Calibration Data Entry** window, enter the calibration data for the tags that you selected.
 6. To navigate among the tags that you selected for error calculation, do one of the following:
 - On the module toolbar, click  to open the **Tag List** pop-up window, and click the tag for which you want to enter calibration data.
 - On the module toolbar, click **Next**  to display the next tag or click **Previous**  to display the previous tag.

NOTE

You have to enter the calibration data for all the tags that you want to include in a loop error calculation on the same day that you intend to perform the calculation. If you enter tag calibration data more than once on a calculation day, the calculation is performed using the results that deviated the most from the predefined tolerance.

7. Click **Options > Loop Error**.
 8. If a message appears with a list of tag numbers that are not yet calibrated, do the following:
 - a. In the **Calibration Data Entry** window, enter the calibration data for the desired tag numbers.
 - b. Click **Options > Loop Error**.
 - c. On the **Loop Error Calculation Method** message box, select the desired calculation method.
- CAUTION** The calculation method that you set here cannot be changed later. Make sure that you set the optimal calculation method for the selected loop.
- d. Click **OK**.
 9. On the **Loop Error Result** dialog box, after you view the results, click **Close**.

Loop Error Result Dialog Box

This dialog box options enable you to view loop error calculation results. Loop error is calculated at the default point that you define in the **Preferences** dialog box.

Show result at point — Select a point at which you want to measure the loop error.

Window — View the loop error results at the point selected in the **Show result at point** list. You can view loop error results per tag. Also, you can view the entire loop error results. The software displays loop error results per tag in the row where the tag number is displayed. The entire loop error results are displayed in the **Loop Error** row.

Loop — View the loop number whose loop error has been calculated.

Tag — View all the calibrated tags associated with the current loop in the loop error calculation.

Upper Tolerance % — View the upper tolerance percentage of the permissible loop error calculation per tag.

In the **Loop Error** row, view the upper tolerance percentage of the permissible loop error calculation in the entire loop.

Lower Tolerance % — View the lower tolerance percentage of the permissible loop error calculation per tag.

In the **Loop Error** row, view the lower tolerance percentage of the permissible loop error calculation in the entire loop.

As Found — View the pre-calibrated values for each calibration point. Values highlighted in red exceed the permissible error. Values highlighted in light blue exceed the intermediate bounds, but do not exceed the permissible error.

As Left — View the new calibration values for each calibration point. Values highlighted in red exceed the permissible error. Values highlighted in light blue exceed the intermediate bounds, but do not exceed the permissible error.

% As Found Error — This field displays the As Found error depending on your worst calibration result for the current tag within the same time span. View the As Found loop error value per each tag.

In the **Loop Error** row, view the calculated As Found loop error value for the current loop. This value depends on all worst calibration results for the tags associated with the loop.

% As Left Error — This field displays the As Left loop error (% of Span or % of Reading depending on your worst calibration result for the current tag within the same time span.

Working with the Fluke Interface

This interface allows you to exchange calibration data with the Fluke Documenting Process Calibrator models 743B and 744. By means of wizards, you can download (export) tag number data to the calibrator, and after collecting the calibration data, you can upload (import) calibration data into SmartPlant Instrumentation for all of the calibrated tag numbers.

IMPORTANT When performing calibration checks with the Fluke calibrator, discrepancies could arise between the values entered in SmartPlant Instrumentation and the values downloaded to the Fluke calibrator. These discrepancies are caused by differences in the number of decimal places allowed by SmartPlant Instrumentation and the Fluke calibrator resolution causing the calibrator to round up or down downloaded values.

For example, temperatures in the Fluke Documenting Process Calibrator are limited to one tenth of degree Celsius resolution. If the downloaded value has two or more decimal places, the values are rounded by the Fluke calibrator and therefore are different to those in SmartPlant Instrumentation. This causes unexpected deviations from the anticipated point levels that will be reflected in the calibration result points.

For more information on the level of accuracy for the different values measured by your Fluke calibrator, please consult your Fluke documentation or Fluke support.

NOTES

- If you upgraded the software from a version prior to 2007.5, you must restore the standard forms and regenerate the form pages to be able to use existing calibration settings and data. For details, see *Upgrading the Standard Calibration Forms* (on page 400).
- When performing calibrations using the Fluke interface, there are two special forms that come shipped with SmartPlant Instrumentation: *d_ff_default_fluke_result.psr* and *d_ff_default_fluke_settings.psr*. These forms can be found on the hard drive where you installed the application under the 'Program Files\SmartPlant\Instrumentation\PSR' folder. For more information about using and assigning forms, see *Managing Spec Pages Common Tasks*, *Spec Forms Common Tasks*, and *PSR File Customization Using Infomaker*.
- If you are working on a Citrix console to which a Fluke Calibrator is connected, to enable the software to detect the calibrator on the serial port located on the client machine, you must first map the serial port to the client machine. For details, see *Map a Serial Port to Use the Fluke Calibrator on a Citrix Console* (on page 422).

Set up a Profile for the Fluke Calibrator

This procedure describes the main workflow involved in preparing calibration forms for use with the Fluke Calibrator. For general details of the desired, procedure, see *Add Customized Calibration Forms to the Database* (on page 404).

1. In the Specifications module, restore the Calibration settings and data entry forms.
2. Create new calibration pages for Fluke as follows:
 - a. Open the Page Editor.
 - b. Open the file 'd_ff_default_fluke_settings.psr' (located in the path: 'Program Files\SmartPlant\Instrumentation\PSR').
 - c. Regenerate the page.
 - d. Click **Save as Page** .
 - e. On the **Save as Page** dialog box, click **New**.
 - f. Type a name for the page (**Fluke Settings**) and if desired, a description.
 - g. Open the file 'd_ff_default_fluke_result.psr' and repeat the above steps c-e.
 - h. Type a name for the page (**Fluke Data Entry**) and if desired, a description.
3. Using these calibration pages, create new calibration forms for settings and data entry and save them to the database.

NOTE These forms are similar to the standard calibration forms, but with the following additional properties:

- **CALIB_RES_UDF_C01...C20** — Fields for user-defined results (data entry) values.
- **CALIB_SET_UDF_C01...C15** — Fields for user-defined settings values.
- **Loop Power** — **None**, **Enabled 24V**, **Enabled 28V**.
- **Temperature Element Type** — All T/C and RTD supported types.
- **TC Source Mode** — Applicable for T/C element types only.

- **Source Mode** — Corresponds to the **sUEV** Fluke property for source user-entered values. The value can be **Auto** or **Manual Entry**.
 - **Measure Mode** — Corresponds to the **mUEV** Fluke property for measure user-entered values. The value can be **Auto** or **Manual Entry**.
4. Define a new profile for Fluke and assign the Fluke forms that you created. For details, see *Create a Calibration Profile* (on page 405).

Map a Serial Port to Use the Fluke Calibrator on a Citrix Console

1. Within the Citrix console, at the server location, click **Start > Run**.
2. On the **Run** dialog box, in the **Open** box, type `cmd`.
3. At the command prompt, type `net use`.

A summary of the mapping between local and remote ports similar to the following should appear:

```

Command Prompt
Microsoft Windows [Version 5.2.3790]
(C) Copyright 1985-2003 Microsoft Corp.

C:\Documents and Settings>net use
New connections will be remembered.

Status          Local          Remote          Network
-----
A:              \\Client\A$    \\Client\A$    Client Network
C:              \\Client\C$    \\Client\C$    Client Network
COM1            \\Client\COM1: \\Client\COM1: Client Network
D:              \\Client\D$    \\Client\D$    Client Network
LPT1            \\Client\LPT1: \\Client\LPT1: Client Network
LPT2            \\Client\LPT2: \\Client\LPT2: Client Network

The command completed successfully.

C:\Documents and Settings>

```

4. If the desired port, for example COM 1, does not appear in the summary, then type: `net use COM1: \\Client\COM1:`.
5. At the command prompt, type `net use` again and check that the desired serial port is now included in the mapping.

Download Data from SmartPlant Instrumentation

IMPORTANT When performing calibration checks with the Fluke calibrator, discrepancies could arise between the values entered in SmartPlant Instrumentation and the values downloaded to the Fluke calibrator. These discrepancies are caused by differences in the number of decimal places allowed by SmartPlant Instrumentation and the Fluke calibrator resolution causing the calibrator to round up or down downloaded values.

For example, temperatures in the Fluke Documenting Process Calibrator are limited to one tenth of degree Celsius resolution. If the downloaded value has two or more decimal places, the values are rounded by the Fluke calibrator and therefore are different to those in SmartPlant Instrumentation. This causes unexpected deviations from the anticipated point levels that will be reflected in the calibration result points.

For more information on the level of accuracy for the different values measured by your Fluke calibrator, please consult your Fluke documentation or Fluke support.

1. Connect the Fluke Documenting Process Calibrator to your computer's COM port and switch on as described in the *Fluke User Manual*.
2. Open the Calibration module and from the **Tag Calibration Settings** window, define the source and output value ranges and units of measure for the desired tag numbers.

NOTE When performing a calibration for a thermocouple using the Fluke calibrator as an external power supply, you must enter the following data using the custom fields indicated:

Custom Field	Property	Acceptable Value
CALIB_SET_UDF_C1 6	Loop power	24V / 28V
CALIB_SET_UDF_C1 7	Thermocouple type	E / N / J / K / T / B / R / S / C / L / U
CALIB_SET_UDF_C1 8	Thermocouple source mode	Linear MV / Linear T

3. Click **Actions > Fluke Interface**.
4. Click **Next**.
5. On the **Connection Options** page, define your connection setting and click **Test**.
NOTE If the connection test is successful, an appropriate message appears in the **Connection result** box. If the connection fails, check the physical connections and if necessary, select a different communication port and retry.
6. Click **Next** to go to the **Activity Selector** page.
NOTE If you first want to clear the calibrator, click **Flush calibrator memory**, ensure that the **Upload default memlist and procedures** check box is selected, and then click **Next**. On completion of the process click **Back**.
7. Click **Download data from SmartPlant Instrumentation to the calibrator**.
8. On the **Tag Selection for Download** page, select the **Export** check box beside each tag that you want to download to the calibrator, and then click **Next**.
9. On the **Download Data** page, click **Start** to begin the download.
TIP The progress bars indicate the current activity. You will also see an indication on the calibrator that data is uploading to it.

NOTE This procedure downloads the following data to the Fluke calibrator:

- Tag number.
- Component_ID value — this is a unique value that SmartPlant Instrumentation automatically generates on creation of a tag number. This number is displayed in the Fluke calibrator interface as the tag serial number. **Do not modify this number**, either from the calibrator or from the SmartPlant Instrumentation database.
- Source maximum and minimum range values and their units of measure.
- Output measurement maximum and minimum range values and their units of measure.
- Calibration strategy.
- Tolerance.

The following table shows a sample set of downloaded data values:

Tag Number	Range Min.	Range Max.	Range UOM	Output Signal Min.	Output Signal Max.	Output Signal UOM	Strategy (Calibration type)	Tolerance
101- TT - 100	100	200	°C	4	20	mA	3 ↑↓	1

Tag Number	Range Min.	Range Max.	Range UOM	Output Signal Min.	Output Signal Max.	Output Signal UOM	Strategy (Calibration type)	Tolerance
101- PT - 200	0	10	Bar (G)	4	20	mA	5 ↑	1.5
101- LT - 300	0	1000	mm	4	20	mA	3 ↑	2

Upload Data from the Fluke Calibrator

1. After calibrating your instruments, connect the Fluke Documenting Process Calibrator to your computer's COM port and switch on as described in the *Fluke User Manual*.
2. Open the Calibration module.
3. Click **Actions > Fluke Interface**.
4. Click **Next**.
5. On the **Connection Options** page, define your connection setting and click **Test**.
NOTE If the connection test is successful, an appropriate message appears in the **Connection result** box. If the connection fails, check the physical connections and if necessary, select a different communication port and retry.
6. Click **Next** to go to the **Activity Selector** page.
7. Click **Upload data into SmartPlant Instrumentation from the calibrator**.
8. Click **Next**.
9. On the **Upload Data** page, click **Start** to begin uploading data from the calibrator.
TIP The progress bars indicate the current activity. You will also see an indication on the calibrator that data is downloading from it.
10. On completion of the upload process, click **Next**.
11. On the **Upload Summary** page, view the tags for which the interface has uploaded calibration data into SmartPlant Instrumentation.
12. Click **Finish** to complete the operation.
13. In SmartPlant Instrumentation, open the Calibration module and from the **Calibration History** window, view the tag calibration data.

NOTES

- The following pressure units of measure are supported by SmartPlant Instrumentation when working with the Fluke 743B or 744 Calibrators:
 - Psi
 - Bar
 - InHg
 - mBar
 - inH2O
 - mH2O
 - ftH2O
 - Pa
 - kPa
 - gf/cm² (Fluke: g/cm²)

- Any other pressure unit of measure not supported by the Fluke Calibrator (as 'base' units) will default to 'psi' when downloaded to the calibrator.
- Other units of measure that are not supported by the Fluke Calibrator will appear as voltage units.

Index

A

- Access Rights in the Maintenance Module • 351
- Activate the Construction Module • 384
- Add a Block to a Block Type • 259
- Add a Block Type • 259
- Add a Cable Set • 31
- Add a Cell to the Cell Library (MicroStation) • 314
- Add a Connector to a Switch • 246
- Add a Connector to an Amplifier • 243
- Add a Group of Terminals to a Terminal Strip • 87
- Add a Port to a Hub • 251
- Add a Switch Port • 246
- Add a Terminal to a Terminal Strip • 86
- Add a Wire to a Cable Set • 32
- Add an Amplifier Channel to an Amplifier • 243
- Add an Apparatus to an Apparatus Group • 76
- Add an Installation Index • 389
- Add an Installation Index Category • 390
- Add an Intercom Connector • 252
- Add Cables and Objects to an Existing Cable Block Diagram • 160
- Add Customized Calibration Forms to the Database • 404
- Add Document References in Hook-Up Drawings • 337
- Add Intercom Equipment • 252
- Add Jumpers to the Existing Wiring • 98
- Add User-Defined Wiring Equipment Category Properties • 63
- Additional Settings • 366
- Apparatus • 73
- Append Routing Data to a Cable • 151
- Apply a Drawing Generation Method to Hook-Ups • 335
- Apply a Reference Cable to Multiple Cables Without Sets and Wires • 33
- Apply a Reference Cable to One Cable Without Sets and Wires • 34
- Apply Loop Drawing Generation Method • 283
- Apply Naming Conventions to Wiring Items • 22
- Apply Regional Wire Colors to Project Cables • 35
- Arrangement of Telecom Items in the Domain Explorer • 230
- Assign a Segment to a Channel • 108
- Assign an Enhanced Symbol to a Hook-Up • 338
- Assign an Instrument Block to a Drawing Page • 267
- Assign an Instrument to a Channel • 107
- Assign Border and Logo Blocks to Hook-Up Drawings • 337
- Assign Cables to Drums Automatically (with Optimization) • 155
- Assign Cables to Drums Manually (Without Optimization) • 156
- Assign Connector Pins to Cable Wires • 201
- Assign Dimensional Data Forms • 369
- Assign General Blocks to All Loop Drawings via Instrument Type • 268
- Assign Installation Indexes to Items • 390
- Assign the Same Active Item Library to More Than One Plant • 320
- Assigning an External Macro Data Source • 307
- Assigning Wire Designation • 99
- Associate a Cable with a Panel • 161
- Associate a Loop Block • 263
- Associate a Pin or a Terminal with a Channel • 201
- Associate a PLC or DCS Panel with a Profibus DP Segment • 215
- Associate a Routing Section with Cables • 149
- Associate a Segment with a Home-Run Cable • 203
- Associate a Symbol with an Item • 23
- Associate a Symbol with Multiple Items • 23
- Associate a Telephone Number with a Channel • 248
- Associate an I/O card with an I/O Termination • 72
- Associate an Instrument Type with a DDP Group • 369
- Associate an Instrument with a Segment • 183
- Associate Cables with a Pulling Area • 154
- Associate Calibration Profiles with Instrument Types • 405
- Associate Function Blocks with Instrument Types • 173

- Associate Function Blocks with Instruments • 179
- Associate Hook-Up Types with Instrument Types • 327
- Associate Instrument Blocks Automatically (via Instrument Type) • 263
- Associate Instrument Blocks Manually • 265
- Associate Instrument Tag Numbers with Hook-Ups • 327
- Associate Instruments with Device Panels • 57
- Associate Items with Hook-Ups • 329
- Associate Items with User-Defined Sub-Libraries • 322
- Associate Profibus DP Instruments with a DP Segment • 214
- Associate Profibus PA Instruments with a PA Segment • 214
- Associating an External CAD Drawing • 275
- Automatically Propagate a Signal from a Non-Wiring Instrument • 136
- Auto-Wire Control Systems • 128
- Auto-Wire Control Systems Connected to Pre-Assigned Junction Boxes • 130
- Auto-Wire Pre-Assigned Junction Boxes • 126
- Auto-Wire Two Panels • 127
- Auto-Wiring • 123
- Auto-Wiring Task Statuses • 131
- B**
- Block Types • 254
- C**
- Cable Block Diagrams • 159
- Cable Drums • 153
- Cable Routing • 145
- Cable Routing Flow of Activities • 145
- Cables, Cable Sets, and Wires • 25
- CAD Drawing Block Assignment Common Tasks • 261
- CAD Drawing Blocks in SmartPlant Instrumentation • 253
- CAD Loop Drawing Generation • 276
- CAD Loop Drawing Generation Common Tasks • 282
- CAD Settings for SmartPlant Instrumentation • 278
- Calculate Intrinsic Safety • 142
- Calculate Intrinsic Safety for a Loop • 144
- Calculate Loop Errors • 419
- Calculating the Total of Hook-Up Items for a Bill of Material • 346
- Calibration Data Entry Activities • 411
- Calibration Error Calculations • 417
- Calibration Forms • 400
- Calibration Module Commands • 396
- Calibration Module Window • 396
- Calibration Preliminary Activities • 403
- Calibration Settings Activities • 408
- Cancel I/O Assignment • 112
- Change a Hook-Ups Sub-Library Association • 328
- Change an Auto-Wiring Task Status • 132
- Change Instrument / Segment Association • 183
- Change the Block Assignment Method from Automatic to Manual • 264
- Change the Fieldbus Tag Number List Browser View for Segments • 184
- Change the Font Size of Cable and Cable Set Names • 103
- Change the Location of a Panel • 158
- Clear Previous Macro Result Records • 286
- Complete a Maintenance Event Record • 365
- Completing the Plug-and-Socket Group Wizard (Plug-and-Socket Group Wizard) • 78
- Configure a Fieldbus Apparatus • 197
- Connect a Cable or a Cable Set to a Terminal Strip • 94
- Connect a Home-Run Cable to a Plug-and-Socket Box • 202
- Connect a Wire to a Terminal Strip • 96
- Connect Device Cables to a Terminal Strip in Batch Mode • 95
- Connecting to an External Data Source • 305
- Connecting to an ODBC Profile • 306
- Connecting to Database Platforms Other Than ODBC • 306
- Connection Type Examples • 93
- Construction - Example Scenario • 387
- Construction Flow of Activities • 384
- Construction Module • 383
- Construction Module Revisions • 393
- Construction Options • 387
- Control System Tag Propagation • 113
- Conventional and Fieldbus Multi-Input Devices • 221
- Conventional Panels • 37
- Copy Associated Blocks to Other Instrument Tags • 266
- Copy Automatically Assigned Blocks to Tags of Other Loops • 266
- Copy Blocks to Another <Plant> • 261

- Copy Default Data to Working Data • 373
- Copy Existing Process Data • 410
- Copy from Vendor Data to Working Data • 374
- Copy Routing Data to Another Cable • 150
- Copy Working Data to the Default Library • 373
- Couple an Instrument with a Control System Tag • 110
- Create a Cabinet • 39
- Create a Cable • 28
- Create a Cable Block Diagram • 159
- Create a Cable Without Sets and Wires • 33
- Create a Calibration Profile • 405
- Create a Cell Library (MicroStation) • 313
- Create a Channel • 108
- Create a Connector • 29, 200
- Create a Control System Tag • 109
- Create a Control System Tag in the Instrument Index Standard Browser View • 109
- Create a Cross Wiring Cable • 122
- Create a DCS (Distributed Control System) Panel • 42
- Create a Device Cable • 58
- Create a Distribution Frame • 238
- Create a DP-Compatible PLC or DCS Panel • 215
- Create a Fieldbus Brick Assembly • 195
- Create a Fieldbus I/O Card • 205
- Create a Foundation Fieldbus Segment • 182
- Create a General Signal • 138
- Create a HART Instrument • 218
- Create a Home-Run Cable • 185
- Create a Hook-Up Item • 321
- Create a Hook-Up Type • 326
- Create a Hub Cabinet • 249
- Create a Hub Connector • 250
- Create a Junction Box • 37
- Create a Local Tag Signal • 137
- Create a Maintenance Event Record • 364
- Create a Marshaling Rack • 38
- Create a Miscellaneous Cabinet • 248
- Create a Multi-Input Device Panel with Terminal Connections • 223
- Create a Multi-Input Instrument with Plug-and-Socket Connections for a Fieldbus Segment • 223
- Create a Multi-Input Instrument with Plug-and-Socket Connections for a Serial Loop Interface • 226
- Create a Multi-Input Instrument with Terminal Connections for a Fieldbus Segment • 225
- Create a Multi-Input Instrument with Terminal Connections for a Serial Loop Interface • 227
- Create a New Hook-Up Macro Definition • 298, 344
- Create a New Repair Form • 361
- Create a PA Cabinet • 241
- Create a PABX Cabinet • 244
- Create a PABX Rack • 245
- Create a Panel with Terminals and Connectors • 44
- Create a Patch Panel • 239
- Create a PLC (Programmable Logic Controller) Panel • 43
- Create a Plug-and-Socket Device Panel • 187
- Create a Plug-and-Socket Junction Box • 188
- Create a Plug-and-Socket Multi-Input Device Panel • 222
- Create a Profibus DP Segment • 214
- Create a Profibus PA Segment • 214
- Create a Rack • 46
- Create a Routing Section • 148
- Create a Slot • 47
- Create a Splice Panel • 238
- Create a Spur Cable • 186
- Create a Switch • 245
- Create a Switch Channel • 247
- Create a Telecom Cable • 236
- Create a Telecom Conventional Field Device • 235
- Create a Telecom Field Tag • 234
- Create a Telecom Junction Box • 237
- Create a Telecom Plug-and-Socket Field Device • 236
- Create a Terminal Strip • 85
- Create a Test Equipment Tag • 408
- Create a User-Defined Function Block • 172
- Create a User-Defined Macro Function • 302
- Create a Virtual Digital Tag • 220
- Create a Wiring Equipment Item • 64
- Create an Amplifier • 242
- Create an Apparatus Group • 75
- Create an External Terminator in a Plug-and-Socket Box • 192
- Create an External Terminator on a Terminal Strip • 192
- Create an I/O Card • 68
- Create an I/O Card Compatible with HART Instruments • 220
- Create an I/O Termination • 70
- Create an Instrument with a Manually Configured Device Panel • 52
- Create an Instrument with an Automatically Configured Device Panel • 51

- Create an Instrument Without a Device Panel • 54
- Create an Intercom Cabinet • 251
- Create an Internal Terminator • 192
- Create an Intrinsically Safe Circuit Type • 141
- Create an Item Library • 319
- Create an Unassociated Conventional Device Panel • 41
- Create an Unassociated Telephone Number • 248
- Create and Associate a Telephone Number • 247
- Create and Manage a Pulling Area • 153
- Create Device Panels and Cables in the Instrument Index Standard Browser View • 55
- Create Device Panels for Selected Instruments • 55
- Create Foundation Fieldbus and Profibus Instruments • 176
- Create Hook-Up Items in Batch Mode • 321
- Create Hook-Ups • 326
- Create Hub Equipment • 250
- Create Tag Sets (MicroStation) • 311
- Create User-Defined Sub-Libraries • 320
- Creating a Wiring Scheme • 21
- Creating and Managing Cables • 25
- Creating Wiring Equipment for Cable Block Diagrams • 162
- Cross Wiring • 117
- Customize and Use Wiring Equipment Categories • 62
- Customize Cable Routing Terminology • 147
- Customize the Controller/Processor List Header in I/O Card Properties • 71
- Customize the Default Library Display and Layout • 377
- Customizing Macro Definitions • 297
- D**
- Decouple an Instrument and a Control System Tag • 110
- Define a Cable Configuration • 26
- Define a Connection Type • 91
- Define a Dimensional Group • 368
- Define a Drawing as a Multi-Drawing • 271
- Define a Drawing as a Multi-Page Drawing • 272
- Define a New Panel Location • 157
- Define a Preventive Maintenance Activity • 354
- Define a Preventive Maintenance Attachment • 353
- Define a Preventive Maintenance Task • 354
- Define a Process Connection End Preparation • 367
- Define a Terminal Strip Configuration • 81
- Define Alarm / Trip Settings • 410
- Define an Apparatus Configuration • 73
- Define Cable Drum Attributes for Cable Types • 154
- Define Calibration Profile Settings • 405
- Define Circuit Cables for Intrinsic Safety Loop Calculation • 143
- Define Dimensional Group Properties • 368
- Define Foundation Fieldbus and Profibus Instrument Type Profiles • 176
- Define Insertion Points for Blocks • 260
- Define Instrument Type Profiles for HART Instruments • 218
- Define or Edit a Connector Type • 199
- Define or Edit a Connector Type Pin Configuration • 200
- Define Plug-and-Socket Group Layout (Plug-and-Socket Group Wizard) • 77
- Define Segment-Wide Parameters • 170
- Define Tag Calibration Settings • 409
- Define Tag Numbers for Intrinsic Safety Loop Calculation • 143
- Define the Cell Origin (MicroStation) • 314
- Define the Coordinates of the Cell Origin (MicroStation) • 314
- Define the New Ports Group (Plug-and-Socket Group Wizard) • 77
- Define the Plug-and-Socket Ports (Plug-and-Socket Group Wizard) • 78
- Define the Ports (Plug-and-Socket Group Wizard) • 78
- Define the Process Connection Class/Rating • 367
- Define the Set Point • 410
- Defining and Executing Auto-Wiring Tasks • 123
- Defining and Managing Maintenance Event Records • 363
- Defining Test Equipment • 407
- Definitions in the Construction Module • 385
- Delete a Cable Configuration • 28
- Delete a Connector • 202
- Delete a Connector Type • 200
- Delete a Control System Tag • 111
- Delete a Fieldbus Segment • 184
- Delete a Hook-Up Macro Definition • 298, 344
- Delete a Panel Location • 158
- Delete a User-Defined Macro Function • 303
- Delete an Installation Index • 386

- Delete CAD Loop Drawing Generation History • 286
 - Delete Hook-Up Items • 323
 - Delete Hook-Up Sub-Libraries • 324
 - Delete Hook-Up Types • 331
 - Delete Hook-Ups • 331
 - De-Multiplex Tag Signals • 140
 - Device Panels and Cables • 49
 - Device Panels in SmartPlant Instrumentation • 49
 - Dimensional Data and Piping • 365
 - Disconnect Cables from Items in a Cable Block Diagram • 161
 - Disconnect Cables, Cable Sets, and Wires • 97
 - Display a Cell on the Desktop (MicroStation) • 315
 - Display Adjacent Connections of a Cable • 100
 - Display and Access Vendor Data • 370
 - Display and Access Working Data • 372
 - Display AutoCAD Hook-Up Drawings with the Enhanced Report Utility • 343
 - Display CAD Loop Drawing Generation History • 285
 - Display MicroStation Hook-Up Drawings with the Enhanced Report Utility • 343
 - Display SmartSketch Hook-Up Drawings with the Enhanced Report Utility • 342
 - Display the Farthest Connection of a Cable • 101
 - Displaying CAD Hook-Up Drawings with the Enhanced Report Utility Common Tasks • 342
 - Dissociate a Panel from a Location • 158
 - Dissociate an Instrument from a Segment • 183
 - Dissociate Blocks from Instruments • 267
 - Dissociate Device Panels from Instruments • 57
 - Dissociate Items from Hook-Ups • 330
 - Dissociate Profiles from Instrument Types • 406
 - Dissociate Tag Numbers from Hook-Ups • 330
 - Download Data from SmartPlant Instrumentation • 422
 - Duplicate a Dimensional Group • 369
 - Duplicate a Telecom Field Tag • 234
- E**
- Edit a Cable Configuration • 27
 - Edit a Cable Routing • 150
 - Edit a Routing Section • 149
 - Edit an Archived Revision • 395
 - Edit an Installation Index • 386
 - Edit an Installation Index Category • 385
 - Edit Apparatus Terminal Properties • 76
 - Edit Cable Properties • 30
 - Edit Default Dimensional Data • 377
 - Edit Hook-Up Drawing Identifying Fields • 335
 - Edit the Properties of a Foundation Fieldbus Segment • 182
 - Edit the Properties of a Plug-and-Socket Device Panel • 188
 - Edit the Properties of a Plug-and-Socket Junction Box • 189
 - Edit the Properties of a Terminal • 87
 - Edit Wire Properties • 34
 - Edit Working Data • 372
 - Effect Fieldbus I/O Assignment • 206
 - Effect I/O Assignment for a Segment • 107
 - Effect I/O Assignment for an Instrument • 106
 - Effect I/O Assignment for HART Instruments • 220
 - Enable a Function Block for I/O Assignment • 184
 - Enter Calibration Data • 412
 - Enter Default Data into the Default Library • 376
 - Enter Power Supply Data for Panels and Instrument Tags • 45
 - Enter Preventive Maintenance Details • 358
 - Enter Tag Number Dialog Box • 397
 - Enter Vendor Data • 371
 - Equipment Cabinets and Equipment Items • 241
 - Examples of Farthest Connections of Cables • 101
 - Export Tag Sets (MicroStation) • 312
 - Extend Existing Scheduling • 357
 - External Macro Naming Conventions • 304
 - External Macro Structure Conventions • 304
- F**
- Fence Elements for a Cell (MicroStation) • 313
 - Fieldbus and Profibus Segments Common Tasks • 181
 - Fieldbus Bricks • 194
 - Fieldbus Cables Common Tasks • 185
 - Fieldbus Instruments Common Tasks • 175
 - Fieldbus Plug-and-Socket Boxes (Panels) • 186
 - Fieldbus Validation • 207
 - Fieldbus Validation Common Tasks • 209

- Fill Out a Repair Form • 362
- Filter Maintenance Activities • 352
- Filter the Display of Data in the I/O Assignment Window • 111
- Filter the Local Signal Dialog Box • 137
- Filtering Items on a Cable Block Diagram • 163
- Find I/O Channels for Assignment • 115
- Flow of Activities for Auto-Wiring • 124
- Flow of Activities for Foundation Fieldbus Design • 166
- Flow of Activities for HART Instrumentation • 217
- Flow of Activities for Profibus Design • 211
- Flow of Activities for Telecom Design • 229
- Force Signal Propagation • 135
- Foundation Fieldbus and Profibus • 165
- Foundation Fieldbus Design • 165

G

- General Wiring Definitions • 19
- Generate a Bill of Material • 348
- Generate a Connection Report from the Connection Window • 103
- Generate a Dimensional Data Sheet • 378
- Generate a Group List, Dimensions List, Default Library List, Default Dimensional Data, Or Vendor Dimensional Data Report • 379
- Generate a Hook-Up Item List Report • 349
- Generate a Hook-Up Tag List Report • 349
- Generate a Macro Report • 295
- Generate a Maintenance Report • 353
- Generate a Panel Strip Report from the Cross Wiring Window • 122
- Generate a Report of CAD Loop Drawing Generation Errors • 287
- Generate a Segment Wiring Report • 210
- Generate A Suspected Data List, Status History, Or Suspected Data History Report • 380
- Generate a Validation Report for Multiple Segments • 210
- Generate a Validation Report for Multiple Segments from the Fieldbus Tag Numbers Browser • 210
- Generate a Validation Report for One Segment • 209
- Generate an Enhanced Profibus Layout Report • 216
- Generate an I/O Tag Assignment Report • 112
- Generate CAD Hook-Up Drawings • 339

- Generate CAD Loop Drawings • 283
- Generate Calibration Reports • 416
- Generate DDP Reports • 378
- Generate Enhanced Hook-Up Drawings • 340
- Generate Hook-Up Drawings in Mixed Mode • 341
- Generate Reports For An Installation Index Revision Reports • 393
- Generating a Report Using MicroStation • 315

H

- HART Instruments • 217
- Hook-Up Drawing Generation • 331
- Hook-Up Item Libraries and Sub-Libraries • 317
- Hook-Up Items and Libraries • 316
- Hook-Up Reports • 345
- Hook-Up Reports Common Tasks • 347
- Hook-Ups in SmartPlant Instrumentation • 315
- Hook-Ups Module Window • 316

I

- I/O Assignment • 105
- Import a Tag Set (MicroStation) • 312
- Insert Macros in an AutoCAD Drawing Block • 296
- Installation Index Category Definitions • 385
- Instrument Maintenance • 350
- Intrinsic Safety • 141
- Issue a New Work Request • 360

L

- Link a Tag to a Drawing Element (MicroStation) • 312
- Link Tag Signals to a General Signal • 139
- List of Calibration Reports • 414
- Loop Block Macro Syntax • 289
- Loop Blocks • 256
- Loop Drawing Generation Methods • 277
- Loop Drawing Reports • 286
- Loop Drawing Types and Drawing Properties • 269
- Loop Drawing Types and Drawing Properties Common Tasks • 269
- Loop Drawings Module • 253
- Loop Error Calculation • 416
- Loop Error Result Dialog Box • 420

M

- Maintain CAD Loop Drawing Revisions • 273

- Maintain Document References • 274
 - Maintain Hook-Up Drawing Revisions • 336
 - Maintaining a Revision Archive • 394
 - Maintenance Event Records • 362
 - Make Approvals for Hook-Up Drawings • 336
 - Make CAD Drawing Approvals • 274
 - Making Connections and Associations for Fieldbus • 199
 - Making Connections in SmartPlant Instrumentation • 91
 - Manage Cable Drums Manually • 154
 - Manage Code Tables • 406
 - Manage Default Dimensional Data • 376
 - Manage Intrinsic Safety Circuit Types • 141
 - Manage Standard Positions • 147
 - Manage Standard Widths • 147
 - Manage the Function Blocks Supporting Table • 174
 - Managing CAD Drawing Blocks Common Tasks • 258
 - Managing Fieldbus Brick Assemblies and Apparatuses Common Tasks • 195
 - Managing Fieldbus I/O Cards and I/O Assignment Common Tasks • 205
 - Managing Fieldbus Panels (Plug-and-Socket Boxes) Common Tasks • 187
 - Managing Hook-Up Drawings Common Tasks • 333
 - Managing Hook-Up Items and Libraries Common Tasks • 318
 - Managing Hook-Ups Common Tasks • 324
 - Managing Multi-Input Devices • 221
 - Managing Panel Locations • 157
 - Managing Terminators Common Tasks • 191
 - Map a Serial Port to Use the Fluke Calibrator on a Citrix Console • 422
 - MicroStation and SmartPlant Instrumentation • 307
 - Modify a Terminal Strip Configuration • 83
 - Modify a User-Defined Macro Function • 302
 - Modify Loop Drawing Identifying Information • 274
 - Modify Panel Location Properties • 157
 - Modify Properties of a Single-Page Drawing • 272
 - Modify the Internal Links of a Plug-and-Socket Group • 79
 - Modify Working Data Status • 375
 - Move a Terminal • 88
 - Move a Terminal Strip to Another Parent Item • 85
 - Move an External Terminator to Another Port in a Plug-and-Socket Box • 193
 - Move Cables, Cable Sets, and Wires • 97
 - Move Hook-Ups to Another Hook-Up Type • 328
 - Multiplex Tag Signals • 139
- N**
- Name and Sequence Uniqueness of Wiring Items • 21
- O**
- Open an Existing Work Request • 361
 - Open Reports by Selecting Document Numbers • 350
 - Opening a Work Request • 360
- P**
- Perform Automatic Cross Wiring • 120
 - Perform Breakdown Maintenance • 359
 - Perform Manual Cross Wiring • 117
 - Perform Preventive Maintenance • 356
 - Perform Semiautomatic Cross Wiring • 118
 - Performing Breakdown Maintenance • 359
 - Performing Calibrations • 395
 - Performing Preventive Maintenance • 355
 - Piping Design Area • 373
 - Possible Cases of Signal Propagation • 133
 - Pre-Assign Junction Boxes to Device Panels • 125
 - Preliminary Definitions for Foundation Fieldbus Common Tasks • 169
 - Preliminary Procedures • 367
 - Prerequisites for Connecting to a dBase ODBC Profile • 305
 - Prerequisites for Creating a Test Equipment Tag • 407
 - Preventive Maintenance Preliminary Activities • 353
 - Preview a Cell (MicroStation) • 314
 - Principles of Creating Device Panels • 50
 - Principles of the DDP Module • 366
 - Principles of Wiring Operations in SmartPlant Instrumentation • 19
 - Print Dimensional Data Sheets From the Dimensional Data for Piping Module Window • 380
 - Print Dimensional Data Sheets From the Working Data Window • 380
 - Print Empty Sheet Forms • 381
 - Printing Dimensional Data Sheets • 380
 - Profibus Design • 211
 - Profibus Design Common Tasks • 213
 - Propagate a Tag Signal Manually • 135

R

Regenerate Dimensional Data Sheets • 376
Remove a Jumper from the Existing Wiring • 99
Remove Items from a User-Defined Sub-Library • 323
Remove Trailing Separators From Wiring Item Names • 24
Rename a Control System Tag • 110
Rename Device Cables in Batch Mode • 59
Reports, Revisions and Archives • 393
Re-Propagate a Tag Signal • 135

S

Sample Loop Drawing (After Generation) • 281
Schedule Maintenance • 356
Search for Tag Numbers in the Calibration Module • 398
Select a Calibration Profile • 412
Select a Graphic for a Terminal Connection • 89
Select Test Equipment • 413
Services and Reports • 378
Set a Date Format for Maintenance Event Record Names • 364
Set an Item Library As the Active Item Library • 320
Set Cable Routing Options • 146
Set Cross Wiring Preferences • 120
Set Default Generation Method for Hook-Up Drawings • 335
Set I/O Assignment Instrument Tag Preferences • 105
Set MicroStation for SmartPlant Instrumentation • 308
Set Preferences for Automatic Naming of Jumpers • 99
Set Tags for Loop Error Calculation • 418
Set Telecom Device Type Profile Defaults • 233
Set the Date Format in CAD Loop Drawings • 271
Set the Maximum Number of Cables for the Standard Positions • 148
Set up a Profile for the Fluke Calibrator • 421
Shipped Block Files • 257
Shipped Enhanced Hook-Ups Symbols • 333
Show the Latest Generated CAD Drawing • 285
Signal Propagation • 133

SmartPlant Instrumentation Detailed Engineering User's Guide • 15
SmartPlant Instrumentation Macro Structure • 288
Sort Data in a Bill of Material • 348
Sort Hook-Up Items in a Sub-Library • 322
Specify Border and Logo Blocks for Loop Drawings • 268
Splitting the Wiring Design Among Several Teams • 19
Spur Cable Lengths Example • 171
Standard Functions • 300
Start the DDP Module • 367
Start the Maintenance Module • 351
Starting the Calibration Module • 397
Steps in Loop Drawing Generation Using a CAD Application • 278
Supporting Tables for Telecom • 231
System Requirements • 308

T

Telecom Design • 229
Telecom Devices and Cables • 233
Telecom Panels • 237
Terminal Strip Configuration Examples • 84
Terminal Strips • 81
Terminal Strips and Terminals • 81
Terminator (Plug-and-Socket Box) - Segment Association • 193
Terminator (Terminal Strip) - Segment Association • 193
The Plug-and-Socket Group Wizard • 77
Title Block Macro List • 289
Trace a Signal in a Point-to-Point Wiring Diagram • 102
Track the Data Status History • 378

U

Update Block Assignments • 265
Update Paths for CAD Drawings and Block Files • 275
Update Paths for Hook-Up Drawings and Block Files • 338
Updating Connection Data • 100
Upgrade Calibration Results • 403
Upgrade the Standard Calibration Forms • 402
Upgrading the Standard Calibration Forms • 400
Upload Data from the Fluke Calibrator • 424
Using External Macro Data Sources • 303
Using Macros in CAD Blocks • 295

V

- Vendor Data • 370
- View a Drawing Block in a CAD Application • 260
- View an Archived Installation Index • 394
- View an Existing Cable Block Diagram • 161
- View an External CAD Drawing • 276
- View and Edit Channel Data • 114
- View and Modify Assigned Installation Indexes • 386
- View Cable Connection Destinations • 100
- View Calibration History • 414
- View Installation Indexes in the Installation Indexes Dialog Box • 388

W

- Welcome to the Plug-and-Socket Group Wizard • 77
- Wiring Equipment • 61
- Wiring Equipment and Apparatus • 61
- Wiring in SmartPlant Instrumentation • 17
- Wiring Items Hierarchy Examples • 65
- Wiring Module Macros - Conventions • 293
- Workflow for Creating Hook-Up Items in the Reference Explorer • 317
- Working Data • 371
- Working with Function Blocks • 172
- Working with Macros in SmartPlant Instrumentation • 287
- Working with MicroStation • 311
- Working with Old Equipment • 232
- Working with Terminators • 189
- Working with the Fieldbus Tag Numbers Browser • 175
- Working with the Fluke Interface • 420
- Working with User-Defined Macro Functions • 299